SIEMENS

SINUMERIK 880 GA2, Software Version 1 Installation Instructions

Installation Guide 01.93 Edition

Service Documentation

SINUMERIK 880 GA2, Software Version 1 Installation Instructions

Installation Guide

Service Documentation

Valid for:

Control	*Software Version*
SINUMERIK 880 GA2 T/M	1

01.93 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 the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
01.93	6ZB5 410-0HG02-0AA0	**A**

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

Preliminary Remarks

Notes for the Reader

The SINUMERIK 880 GA2 Installation Guide is divided into two separate manuals:

- Installation Instructions
- Installation Lists

The ”Installation Instructions” discuss the installation and start-up procedures.

The supplementary manual, which is entitled ”SINUMERIK 880 GA2, Installation Lists”, provides additional aids in the form of lists and detailed information on NC and PLC machine data and setting data.

The installation instructions and lists apply to a specific software version only. A new installation guide is required for each new software version. Old installation guides can be used only in part for new software versions.

Contents

1 Preconditions and Visual Inspection

1.1 Preconditions

The following preconditions must be satisfied prior to commissioning:

- Electrical and mechanical installation of the machine must have been completed and the axes prepared for operation.

These points are to be confirmed by the customer.

- Customer PLC program is operational and pretested.
- Measuring system has been installed and wired as far as the SINUMERIK (see Section 1.2.3).
- Cables are connected to the machine. Cable shields run to the controller neutral point as specified in the Interface Description. Flexible earth wires have been installed (see Section 1.2.2). Check that the earthing concept has been kept to!
- Customer personnel support for work on the interface unit, work on machine, machine operation, customer-generated PLC program and the UMS configured by the customer is guaranteed.

  Recommendation:

  First limit travel ranges (greater clearance distances) by displacing the travel limits ("EMERGENCY STOP" cams).
- The specified machine data must be available.
- Test program must be available for checking machine-specific functions.

1.2 Visual inspection

1.2.1 Notes on handling modules

- Synthetic or rubber soling, and in particular synthetic flooring and carpeting, may produce static charging amounting to several kV in human beings. Integrated circuits are sensitive as far as any high-voltage discharge is concerned.
- Even with the controller disconnected, static charging may cause damage, e.g. short-circuiting across the VCC RAM printed conductors which may corrupt the data in the buffered CMOS RAM memories or burn out the printed conductors.
- It is critical therefore that attention should be paid to the notes below during visual inspection and throughout the entire commissioning process in order to prevent damage as a result of incorrect handling.
- Never touch the printed conductors and components without first discharging on an earthed system part.
- Modules and power supply cables should only be withdrawn or inserted with the controller disconnected.

  Note:

  When replacing modules or in the event of a fault during commissioning, all ICs in sockets must be checked for correct location and fit.

Special notes on handling modules with MOS chips

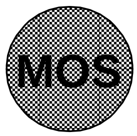
MOS is a technology used to manufacture LSI digital circuits. "MOS" stands for **M**etal-**O**xide **S**emiconductor.

The principal advantages of MOS technology are as follows:

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

Special safety regulations apply to modules with MOS chips. For this reason they are specially marked:

Identification on packaging:



Caution!
Observe safety
regulations!

MOS

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!

Identification on printed circuit board:

M
O
S

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.

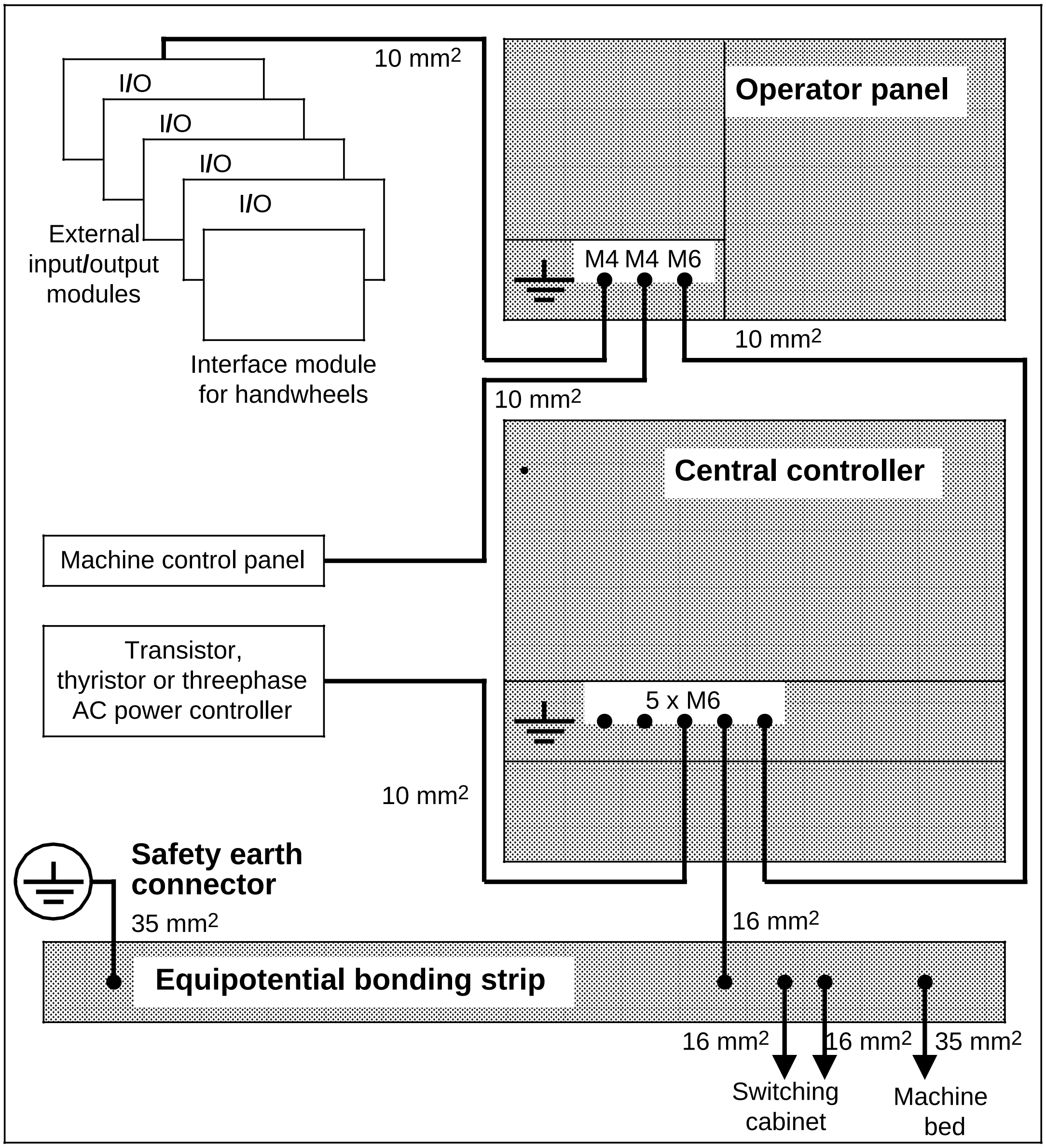
1.2.2 Earthing

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.

Earthing must be carried out as described in the "SINUMERIK 880 Instruction Manual" and the "Distributed Machine I/O Devices Instruction Manual".

Earthing concept (extract from the Instruction Manual)

- Earthing meets the requirements of the DIN VDE 0160.
- The same earthing concept applies to the NC, PLC, drives and machines.
- The earth connections are routed to a central earthing point in a start layout.
- Equipotential bonding of the external components is by means of the equipotential bonding strip.
- Safety earth terminal.



Example of equipotential bonding strip location (earthing concept)

1.2.3 Position encoders

Particular attention is to be paid to the prescribed installation of the graduated scales (air gap etc.) and the pulse encoders (coupling) (see also Installation and Adjustment Instructions of the Manufacturer).

Check for correct wiring and fixed location of the connectors.

If the customer has inserted adapter plugs in the measuring circuit lines, a check must be made for proper connection, strain relief and, above all, the prescribed shielding.

Other makes of position encoder may result in inaccuracy and surface quality problems beyond our control.

See Section 8.1.2 for the characteristics of digital measuring systems with a differential output.

1.2.4 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.

Check that the cables are properly run and that trailing cables are correctly laid. Any kinking must be avoided. Comply with the permissible bending radii. (Also refer to the Interface Description, part 2, for connecting conditions).

1.2.5 Cables

Check all cables in accordance with the cable and equipment overview (see Interface Description, Part 2). This applies particularly to cables produced by the 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.

1.2.6 Shielding

The overall shields of all cables running to or from the controller must be earthed at the controller via the connectors (see Interface Description, Part 2). Only SINUMERIK connectors are permissible because other connectors cause interference.

1.2.7 Interference suppression

All d.c. and a.c. relays must be interference-suppressed using suitable means, as must a.c. or three-phase motors (e.g. lubricating pumps and the like). Observance of the prescribed measures for interference suppression should be checked on a random basis (also refer to the Instruction Manual).

1.2.8 Operator panel

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

It is imperative for operation of the SINUMERIK 880 GA2 that the operator panel is connected to the central controller by means of optical fibres or copper cable. The COM CPU and OP CPU (operator panel CPU) cannot be started up without the connecting cable.

(See Interface Description, Part 2 for cables).

Note:

You must initialize the operator panel when starting up for the first time (see Section 6.1.1)

1.2.9 Overall condition

Check the module mounts and cover plates.

Check that the front panel screws have been tightened (M connection).

Check that the accessory pack is complete.

Check that all unassigned slots have a blanking plate.

The accessory pack must contain the following:

- Log book
- Complete parts list (the parts list is included with the original delivery note and must be inserted in the log book).
- Transparent covers and symbols for customer keys.
- Instruction manual
- Delivery bill with the options ordered.

1.2.10 Batteries

The battery for the NC and PLC data memory (internal PLC) is located on the bottom right-hand side in the central controller. It is accessible from the front for replacement purposes. To this end the cover on the power supply unit must be moved downwards. The battery should only be replaced under power so that the stored data (part programs, setting data, machine data ...) are not lost.

The battery voltage is continuously monitored. Alarm 1 is displayed on the NC screen if the voltage drops below 2.7 V. In this case the control must not be switched off until new batteries have been inserted.

Battery type:

Renewal interval 1 year	3 alkali batteries 1.5 V (baby)

The operator panel has its own battery backup for its internal data. The battery voltage is continuously monitored. Alarm 7 is displayed on the screen if the voltage falls below the tolerance threshold. In this event the operator panel must not be switched off until new batteries have been inserted.

The batteries are accommodated in a compartment on the back of the operator panel.

Battery type:	3 alkali batteries 1.5 V (miniature)

1.3 Export version

The following Options cannot be set on the SINUMERIK 880TE and SINUMERIK 880ME controls:

- > 4D machining
- Interpolation compensation (H57)

2 Installation Checklist

880 M ☐ 880 T ☐ F-No. ____________________

Installation sequence

Section 1 of the Installation Guide, Interface Description Part 2 and the information presented in the Instruction Manual must be carefully observed!

Copy the installation checklist, fill it out, and enclose it in the log book after installation.

Make a cross next to Yes or No after each section has been completed.

Enter all required values where stated.

Information relating to the individual sections is provided in the Installation Guide.

Initial installation

Name .... Office .... Date ....

Manufacturer .... Address ....

Re-installation

Name .... Office .... Date ....

Customer .... Address ....

1. Are the prerequisites for installation per Section 1 fulfilled? Yes ☐ No ☐

2. Visual inspection: Mains connection, EMERGENCY STOP, grounding concept, grounding of the position encoders, cabling, shielding, external machine control panel, logic submodule (I/O submodule), input/output modules, overall state OK? Yes ☐ No ☐

3. Version of the control software:

OP	....
COM	....
NC1	....
NC2	....
Servo1	....
Servo2	....
Servo3	....
Servo4	....
PLC1	....
PLC2	....
UMS	....

4. Voltage test: Voltage at power supply units . . . . (tol. 230 V + 6 %/- 10%)
   Voltage at I/O submodule . . . . (20-30 V, inc. ripple)

5. Standard installation completed and customer-specific machine data entered? Yes ☐ No ☐

6. PLC program entered and tested (safety functions)? Yes ☐ No ☐

7. Position control loops of axes installed and the following checked: Axis speeds / tacho-generator compensation / multgain / servo gain (K_V factor) / acceleration / exact positioning / position control loop monitors / analog spindle speed / spindle positioning / traversing ranges? Yes ☐ No ☐

8. All conventional functions tested? (10 mm programmed = 10 mm on machine) Yes ☐ No ☐

   Function test performed with test program (by customer)? Yes ☐ No ☐

9. Data backup
In the event of a liability case (e.g. exchange of software to remedy errors) SIEMENS AG is not liable for the loss, damage, or reacquisition of lost data.

Were backup copies made of the following data:

• NC machine data	Yes ☐	No ☐
• PLC machine data	Yes ☐	No ☐
• Cycle machine data	Yes ☐	No ☐
• Cycle setting data	Yes ☐	No ☐
• IKA data	Yes ☐	No ☐
• Zero offsets	Yes ☐	No ☐
• Setting data	Yes ☐	No ☐
• R parameters	Yes ☐	No ☐
• Tool offsets	Yes ☐	No ☐
• UMS (as source floppy or source code)	Yes ☐	No ☐
• PLC user program + data blocks	Yes ☐	No ☐
Were these data deposited at the machine?	Yes ☐	No ☐
Installation checklist completely filled out (including options), inserted in the log book and deposited at the control?	Yes ☐	No ☐

10. Were the following functions explained to the customer:

Drift compensation, reference point adjustment	Yes ☐	No ☐
Marginal software conditions as per revision bulletin	Yes ☐	No ☐
Did the customer sign the installation report?	Yes ☐	No ☐

Signatures

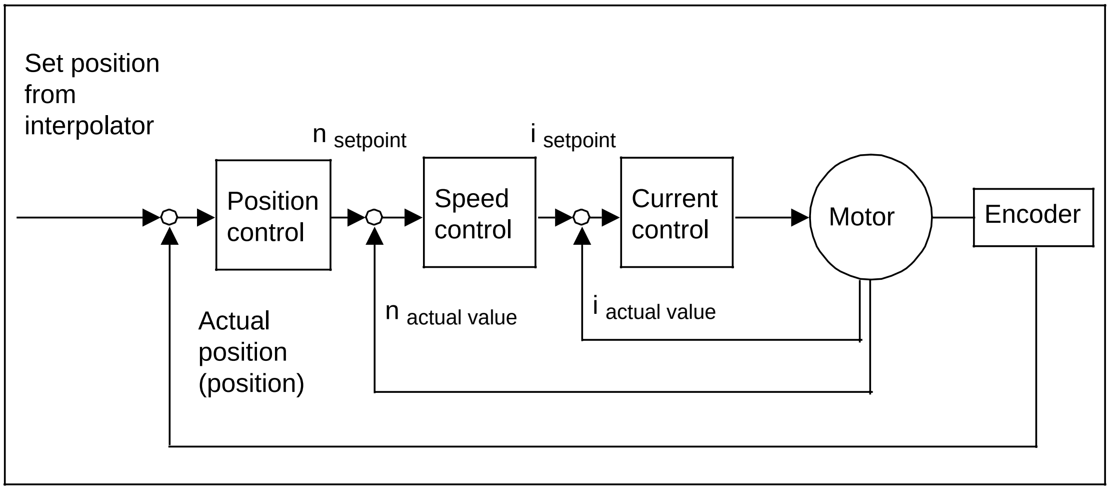
Initial installation

Re-installation

3 Drive Optimization

3.1 Direct control of feed axes - checking and adjustment

Simplified block diagram of the drive control loop



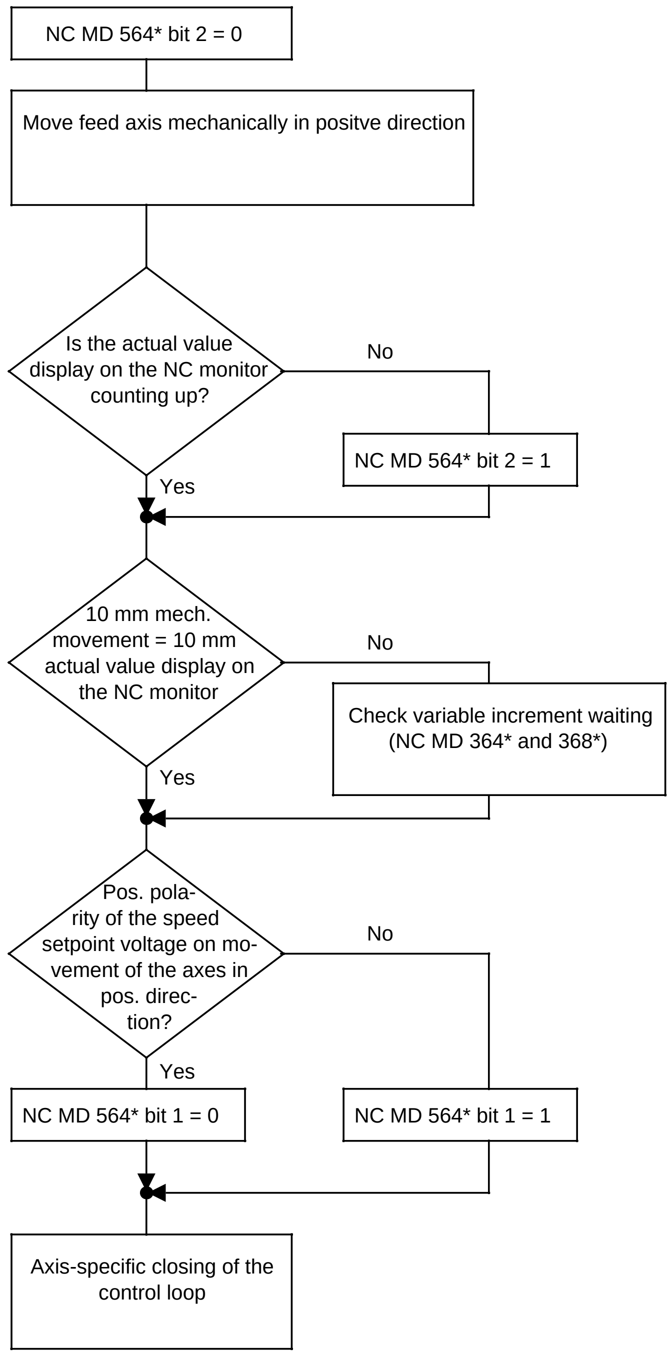
Before the position control is started up the speed control and the current control of the drive must have been started up and optimized.

When you start up the position controller first check the actual position before optimizing the position setpoint.

The following must be determined before starting work:

- The direction of travel of the feed axis (as stated by the customer or according to ISO)
- The polarity of the speed setpoint voltage on the controller for axis movement in positive direction (as stated by the customer or by testing with a battery box)

With NC MD 564* bit 1 (sign inversion speed setpoint) and NC MD 564* bit 2 (sign inversion actual position) you can influence the control direction of the position control (see flowchart).


NC MD 564* bit 2 = 0
Move feed axis mechanically in positve direction
Is the actual value display on the NC monitor counting up?
No
NC MD 564* bit 2 = 1
Yes
10 mm mech. movement = 10 mm actual value display on the NC monitor
No
Check variable increment waiting (NC MD 364* and 368*)
Yes
Pos. polarity of the speed setpoint voltage on movement of the axes in pos. direction?
No
Yes
NC MD 564* bit 1 = 0
NC MD 564* bit 1 = 1
Axis-specific closing of the control loop

3.2 Definition of maximum setpoint and tachogenerator compensation

The axis-specific maximum speeds NC MD 280* to 292* requested by the customer are assigned to a tachogenerator voltage.

It should be borne in mind that a control reserve of roughly 5 % is also required.

The performance limits are set by the measuring-circuit board (10 V) or by the drive control unit.

Definition of maximum setpoint:

The setpoint (command value) is specified under NC MD 268*. In this way the output voltage is limited by the NC. In operation the limit entered under NC MD 268* must not be reached.

The maximum allowable setpoint is 10 V.

Conversion to input units: 10 V ≙ 8191 units

The maximum setpoint must be entered in accordance with the maximum allowable input voltage of the drive control unit.

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

   Input under NC MD 268* : 8191 units

   The maximum axis speed is already reached at 9.5 V on account of the control reserve of 5 %.

b) The drive control unit must be limited to an input voltage of less than 10 V:

   Input under NC MD 268* : < 8191 units

Example

Up to a setpoint of 5 V the value entered is 4100. The maximum axis speed must be reached at 4 V.

In general terms the setpoint should be as high as possible since better control response can be achieved with a higher setpoint voltage.

3.3 Multgain factor NC MD 260*, max. load velocity NC MD 256*

Like the multgain (NC MD 260*), the maximum load velocity (NC MD 256*) serves the purpose of matching the controlled system to the servo gain (K_V) factor (NC MD 252*).

Only the quotient of these two data is used. In the event of accuracy problems or restrictions imposed by the input limits, the factors between NC MD 256* and NC MD 260* can be reduced or enlarged.

Max. load velocity (NC MD 256*) = Velocity [$\frac{mm}{min}$, $\frac{inch}{min}$ or $\frac{deg.}{min}$], derived from the max. motor speed, mechanical gearbox and spindle pitch.

Multgain (NC MD 260*) = Set speed voltage at maximum motor speed [mV].

3.4 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 servo gain value results in instability, overshoot and, possibly, impermissible machine loading.

The maximum allowable servo gain factor depends on:

- Response time constant
- Deadtime of the drive
- Behaviour of the controlled system
- Quality of machine.

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

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

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

Important:

A precondition for correct setting of the K_V 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:

$$0.5\ \text{m/s}^2 \mathrel{\hat{=}} \text{Input } 50$$

The servo gain (K_V) is entered under NC MD 252* using the following conversion equation.

The following applies if K_V factor = 1:

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

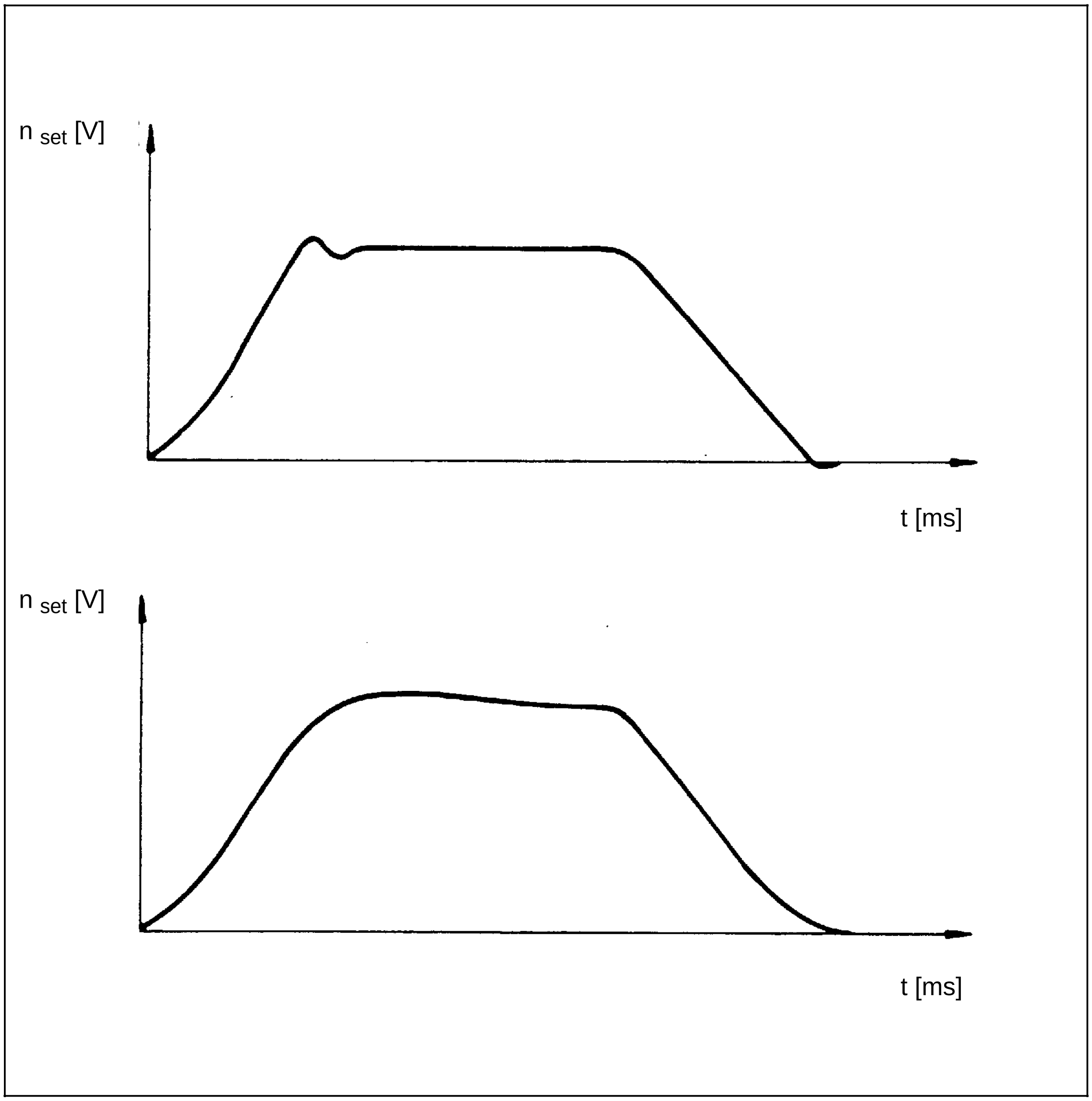
$$= \frac{1000\text{mm/min}}{1\text{mm}} = 1000 \ \frac{1}{60\text{ s}} = 16.66 \ \frac{1}{\text{s}}$$

$$K_V \ (0.01\text{ s}) = 1666 \ \left[\frac{1}{0.01\text{ s}}\right]$$

For servo gain factor (K_V) 1, therefore, the numerical value 1666 is entered.

The least favourable axis in terms of dynamics contributing to continuous-path operation should be assumed to assess whether the positioning behaviour is error-free and whether the set maximum value has been correctly selected.

Setpoint voltage n_{set} with respect to the speed controller is measured using an Oscillomink or storage oscilloscope. Different feedrates are used.



In particular, braking 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.

Checking the 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 (tachogenerator compensation).

3.5 Acceleration NC MD 276*

The axes are accelerated and braked with the input accelerations

Running up to speed and positioning are thus possible with accuracy and speed and with minimum strain on the machine.

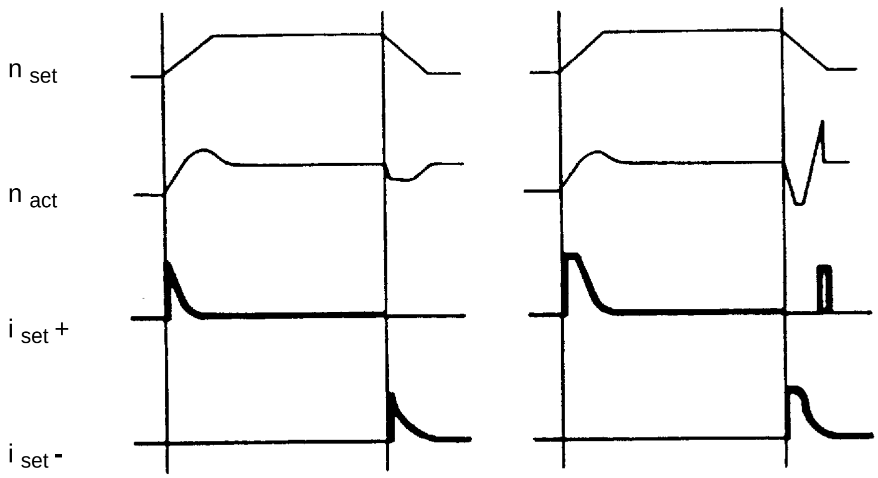
The customer should be consulted to dermine the max. axis acceleration for which the machine is suitable. This value (assuming the drive is not overloaded) is entered under NC MD 276*.

These values are normally 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 speed (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 speed 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. Before the rapid traverse speed or the position are reached, the rpm servo and current control must return to the normal range (current limit **not** reached!).



Example of good acceleration selection

In this example the current limit is reached (can be recognized by the fact that the actual current retains its maximum value over a period of time). **In this case lower acceleration** must be selected.

Relationship between acceleration and actual current

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 may be given different acceleration values, even if they are to interpolate with each other.

3.6 Position monitoring

3.6.1 Exact stop limit coarse and fine (NC MD 204* to 208*)

The position approached is checked. Should the following error be greater than the value entered under NC MD 204* to 208*, then the "MOTION" LED does not go out. The automatic mode is interrupted.

Exception:

The stop tolerance range is not monitored when the axis is clamped.

Setting

Positioning accuracy is dependent on the quality of the position control loop and speed control loop.

The standard deviation is determined by observing the following error at rest.

The input value should be between 10 µm and 50 µm in accordance with the customer's requirements and the positioning accuracy achieved. However, it should be at least twice the maximum deviation of the following error at rest.

3.6.2 Clamping tolerance (NC MD 212*)

The machine manufacturer must endeavour to minimize the position deviation, i.e. as far as possible below the exact stop limit NC MD 204* to 208*. For the clamping tolerance range NC MD 212* the value entered must be approx. twice that under NC MD 204* to 208*. Alarm 112* is displayed when one of the axes at rest (after the time under NC MD 372* has elapsed) is forced out of position (clamping active and servo enable cancelled).

3.6.3 Zero speed control

To make sure that an axis moves into position within the prescribed time (depending on servo gain factor, acceleration, ...), positioning is monitored. This monitoring is set with NC MD 212* and NC MD 372*. The exact description is provided with in NC MD 372*.

3.7 Dynamic contour monitoring

The operating principle of contour monitoring is based on a continuous comparison of the measured following error and that calculated in advance from the NC position partial setpoint. The model used to calculate the following error in advance emulates the dynamic response of the position control including feedforward control.

Dynamic contour monitoring is active not only during the constant velocity phase, but also during acceleration and braking.

However, you must have activated it by setting NC MD 1820* bit 7 = 0 (standard).

A tolerance band for the maximum contour deviation is permitted in order to prevent minor speed fluctuations (caused by load changes) causing erroneous tripping of the monitor.

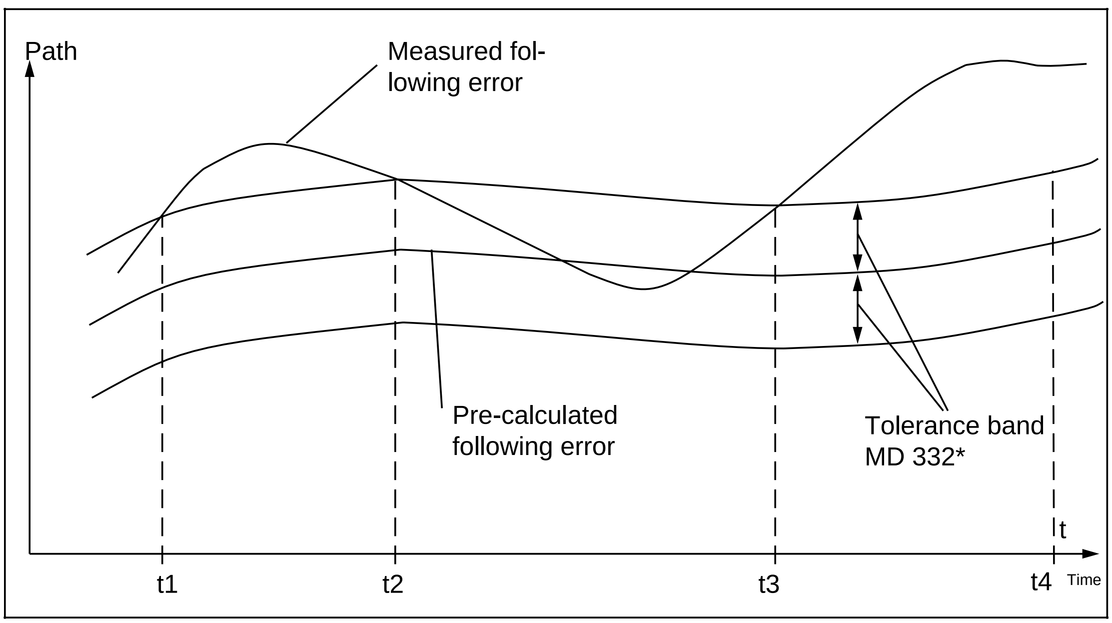
The tolerance band (MD 332*) of the contour monitor is defined as the difference between the real and theoretical following error.

The speed at which the contour monitor is to be active is stated under NC MD 336*. If the value "0" is entered, the contour monitor is active when the axis is stationary as well. When the axes are at rest, the contour monitor also checks for excessive axis movements.

A time value can be entered in NC MD 1200* if you wish to allow the tolerance band for contour monitoring to be exceeded for a short time.

If, in this case, the axis speed is above the response threshold (MD 336*) and the difference between the measured and pre-calculated following error exceeds the tolerance band (MD 332*) for longer than the response time (MD 1200*), the monitor responds.

Example:



$t_2 - t_1 <$ MD 1200*	Control not deselected
$t_4 - t_3 =$ MD 1200*	Control deselected and alarm issued at t_4

The current contour deviations can be displayed for the individual axes in the diagnostics menu under softkey ”SERVICE STATUS”.

When the monitor is triggerd, Alarms 116* are activated and the drives are braked to the current limit by specifying setpoit "0". In addition, the enabling signals for the speed controllers are cancelled and follow-up operation is selected. The alarms can only be cancelled using "RESET" (M2/M30).

4 Overview of Modules and Bus Jumperings

4.1 Overview of modules

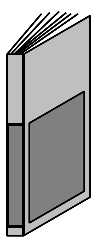
COM area	**Module No.**
COM CPU	6FX1147-4BB00
0.5 MB system EPROM submodule for COM CPU	6FX1124-1CC00
INT/MEM submodule with interfaces and RAM for part programs or similar	6FX1147-5BA01
Memory/page (1-3 memory submodules UMS or similar)	6FX1126-7BA01
256 KB EPROM submodule (standard UMS)	6FX1128-4BC00
128 KB EPROM submodule (for UMS)	6FX1128-4BD00
256 KB EPROM submodule (for UMS)	6FX1128-4BC00
128 KB RAM submodule (for UMS)	6FX1126-6BA00
256 KB RAM submodule (for UMS)	6FX1135-3BA00
Memory/dual-port with clock (single-tier rack)	6FX1124-0BA03
Memory/multi-port with clock (two-tier rack)	6FX1136-8BA01
INT/SINEC H1 CP231 A	6FX1123-1BC02
MEM submodule	6FX1122-6CB00
INT/ACTIVE V.24 (CP315)	6FX1131-5BA01
MEM submodule	6FX1128-4BA00
INT/ACTIVE V24 3*RS232/TTY	6FX1137-3BA01
MAP 3.0 CP1476 ETHERNET H1	6GK1147-6MA00
NC area	
NC CPU	6FX1147-4BB00
0.5 MB system EPROM submodule for NC CPU	6FX1124-1CC00
Servo area	
SERVO-CPU	6FX1136-3BB01
0.5 MB system EPROM submodule for SERVO CPU	6FX1124-1CB00
SERVO interface for 3 measuring circuits	6FX1121-4BA02
SERVO interface for 3 measuring circuits (EXE installation)	6FX1121-4BB02
EXE 5/10-fold	6FX1151-5BA00
HMS interface with 3 inputs	6FX1145-6BB00
HMS interface with 3 inputs and 3 further submodule slots	6FX1145-6BA00
HMS SERVOC command submodule with 1 setpoint	6FX1132-5BA02
Absolute encoder submodule with 3 actual value inputs	6FC9320-4FH
I/V hybrid for linear scale	6FC3988-7CN
Mixed I/O	6FX1138-4BA01

PLC area

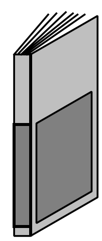
PLC CPU (ACOP 2)	6FX1138-6BL01
256 KB system EPROM submodule for PLC CPU	6FX1145-8BA00
64 KB EPROM submodule (user memory)	6FX1130-5BB00
256 KB EPROM submodule (user memory)	6FX1145-8BA00
INT/DMP (for EUs, DMP)	6FX1144-2BA00
DMP terminal block	6FX1142-1BA01
DMP module 16 inputs/16 outputs	6FX1142-4BA04
DMP module 32 inputs	6FX1142-2BA02
DMP terminator	6FX1145-2BA00
DMP 1P65 terminal block [1]	6FX1152-7BA01
DMP module IP 65 8 inputs + (8 inputs/8 outputs) [1]	6FX1152-8BA02
DMP compact terminal block [1]	6FX1153-2BA00
DMP compact module 8 outputs, 24 V/2.0 A, floating [1]	6FX1153-0BA0
DMP compact module 16 outputs, 24 V/0.5 A, floating [1]	6FX1153-3BA0
DMP compact module 16 inputs, floating [1]	6FX1153-1BA0
INT/CU 16 bit (for PLC 155U)	6FX1137-7BA02
INT/EU 16 bit (for EU 185U)	6FX1137-8BB02

Operator panel area

Power pack 230 V AC	6EW1861-3AD
OP CPU	6FX1120-4BD03
MEM/EPROM-RAM	6FX1128-1BB01
EPROM submodule (for OS or OP software)	6FX1128-4BC00
INT/KEYBOARD	6FX1148-7BA01
INT/OPI	6FX1138-8BC04
INTERFACE	6FX1121-2BA03



Modules other than those stated here can be used in the SINUMERIK 880 GA2. Refer to SINUMERIK 880 GA2 Interface Description, Part 2, Connection Conditions for further information.



4.2 Bus jumpering

A jumper is inserted inside the bus PCB between the first and last CPUs next to each location not carrying a module with link bus plug-in contact. Any jumper missing ahead of the last CPU is not recognized during the start-up routine by the COM CPU and proper operation is no longer possible. These jumpers are correctly inserted at the factory according to the scope of the order (e.g. number of NC CPUs, servo CPUs, PLC CPUs). In the event of a subsequent CPU addition (e.g. installation of a second NC CPU), the jumper next to the module location of the retrofitted CPU has to be removed. Otherwise, the CPU concerned cannot be started.

Refer to SINUMERIK 880 GA2 Interface Description, Part 2, Connection Conditions for further information.

1) *available soon*

5 Voltage and Functional Tests

5.1 Voltage test

5.1.1 Voltage supply

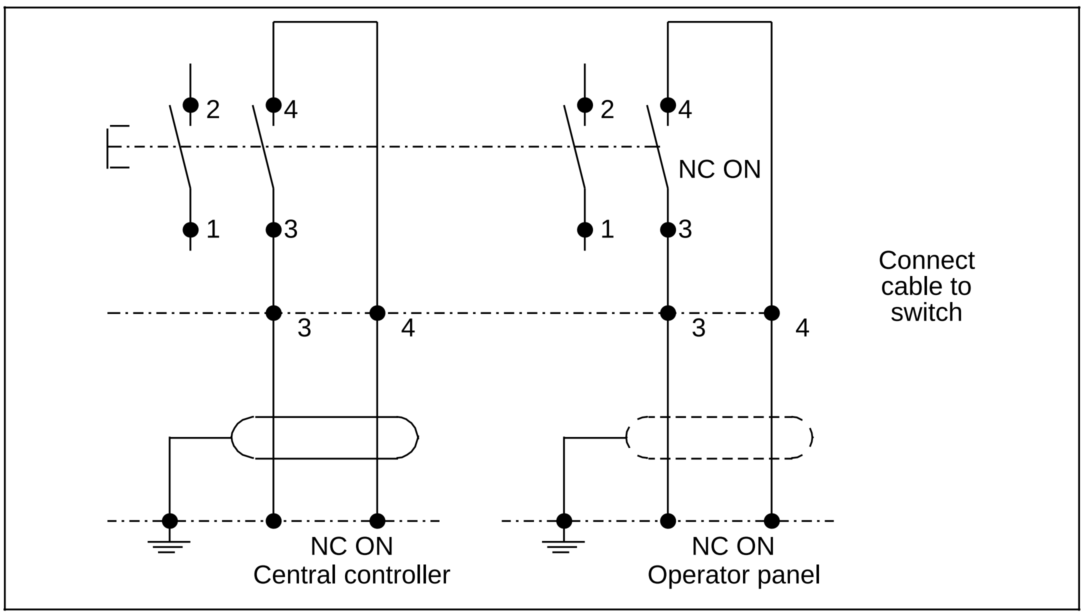
The SINUMERIK 880 GA2 is fitted with two power supply units.

• Operator panel power supply unit	6EW1 861-3AD (5 V/15 A) Voltage: 230 V + 6%, - 10%
• NC power supply unit	6EW1 861-2.. (5 V/40 A) Voltage: 230 V + 6%, -10%

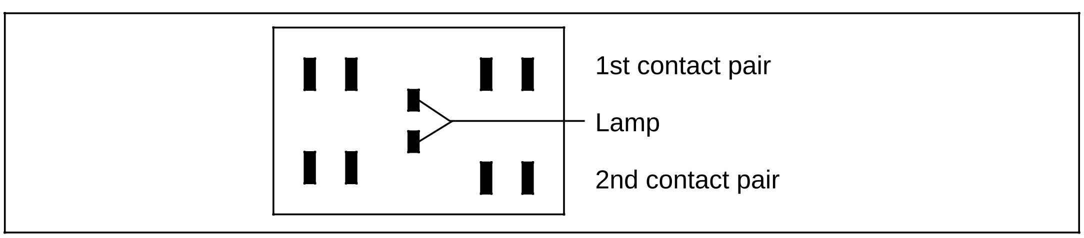
Caution!

NC-ON for the operator power supply unit must **not** be connected in parallel with NC-ON for the central controller power supply unit. Use control switch with 2 contacts.

The connection conditions and voltage supply limit data should be checked **prior** to powering up (see Interface Description Part 2).



Two-contact switch - circuit diagram for NC ON



Switch circuit diagram for NC ON on Siemens machine control panel

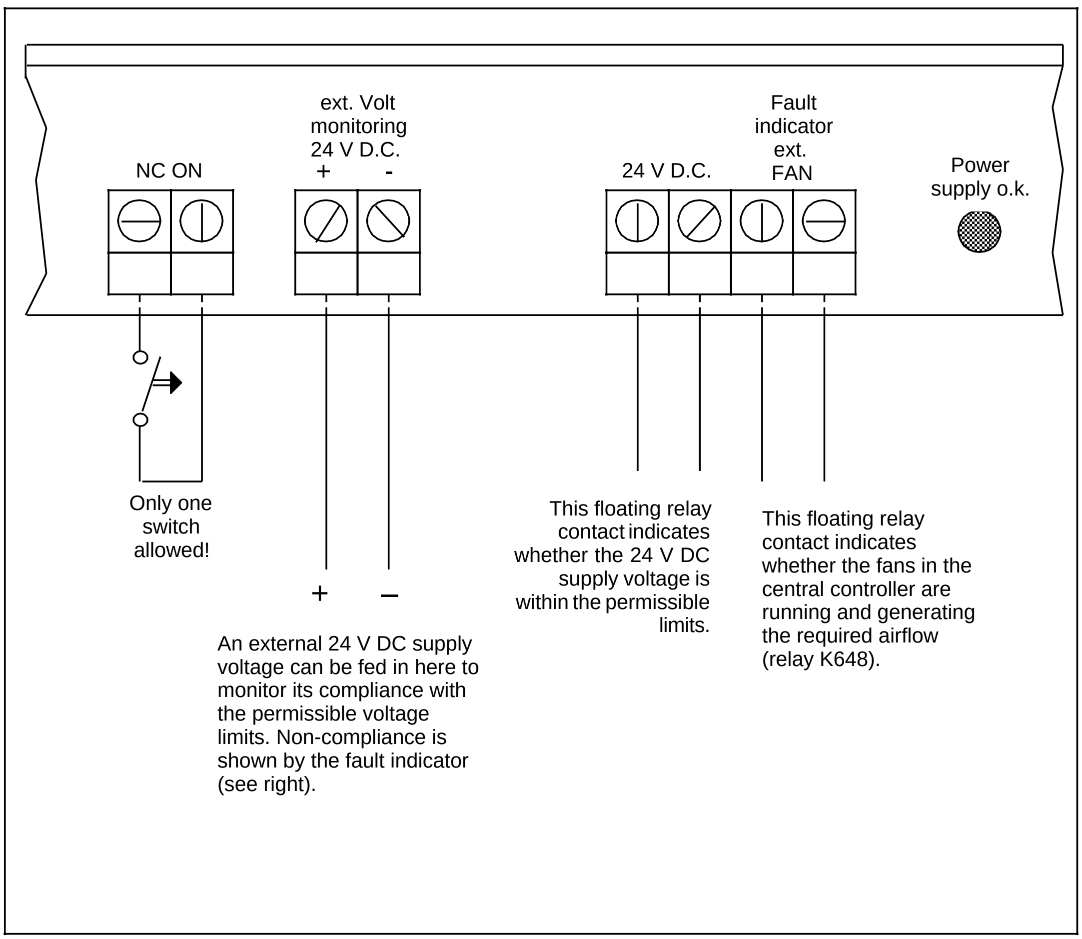
5.1.2 Limit temperature

Ambient temperature 0 ... 55°C

The NC ambient temperature is measured using a temperature sensor on the multiport/dualport and monitored for the upper limit. If the temperature sensor detects an excessive temperature, the control outputs Alarm 2 (overtemperature) on the NC screen.

5.1.3 Fan monitoring

In addition, each of the two fans in the NC power supply unit is monitored by means of one NTC thermistor. Thermistor response need not necessarily result directly in the NC being shut down since any such response only activates relay K648, which the machine manufacturer can then evaluate for himself.



Central controller power supply unit

5.1.4 Direct voltage+ 5V

The 5 V voltage may be measured at all power supply units across the sockets provided and may be adjusted using a potentiometer. The voltage is already set at the factory to approx. + 5.15 V (on account of the line voltage drops), so this voltage is not normally adjusted.

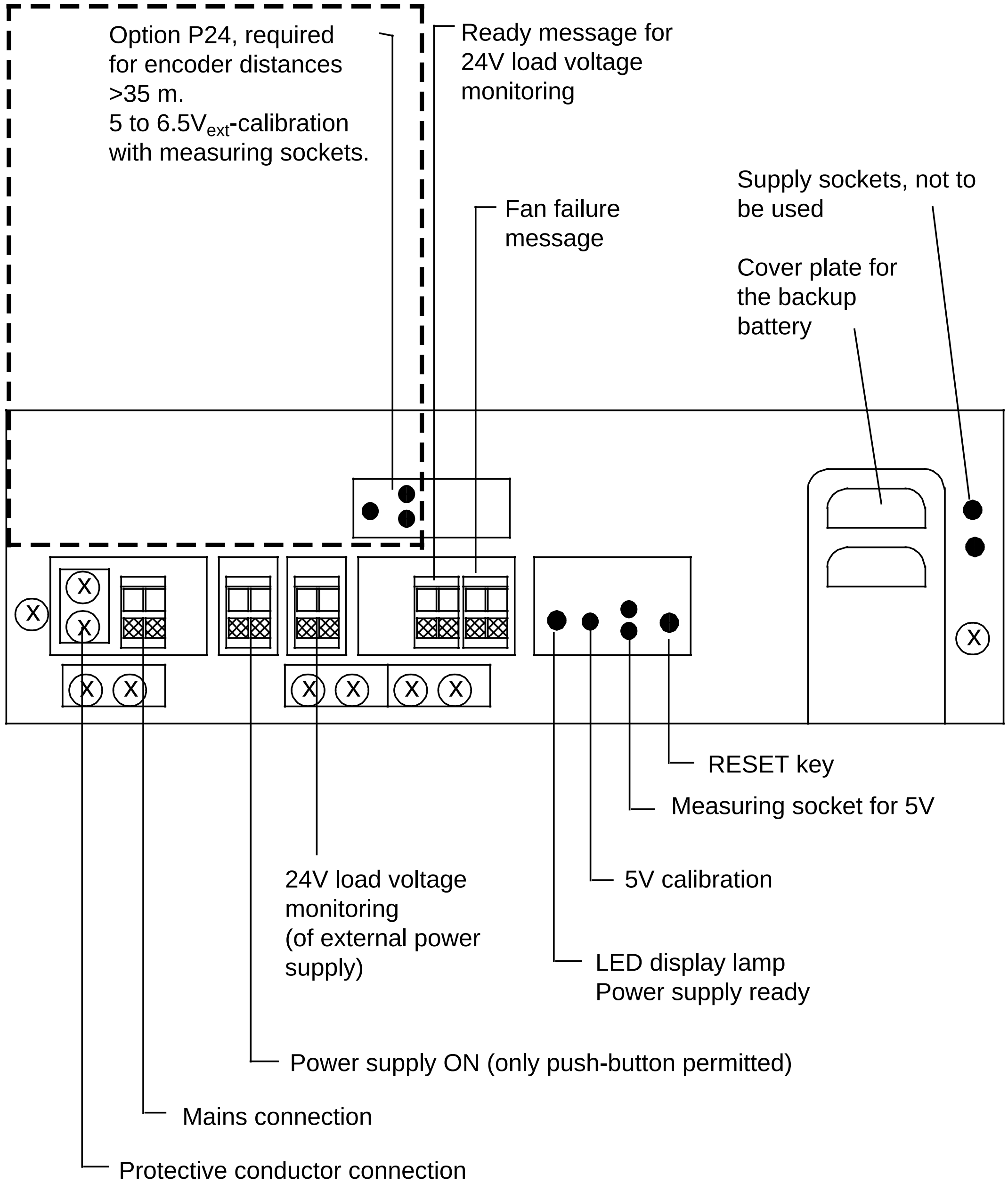
The fault-free operating status is indicated on the NC power supply unit by means of a green LED on the front PCB.

5.1.5 Direct voltage + 5V external

With the 6EW1 861-2B power supply unit (power supply in the 40A central controller) an additional controllable voltage of 5V to 6.5V is available with which encoder distances of more than 35 m are possible. This power supply unit is an Option (Option P24) and can only function with the 6FX1 121-4B measuring circuit modules and the new HMS measuring circuit modules.

For this controllable voltage to be fed from the power supply unit via the measuring circuit modules to the appropriate encoders it is necessary to set jumpers on the measuring circuit modules.

On the power supply unit the adjustable voltage is set as follows:



5.2 Functional test

5.2.1 CPU monitor

The LEDs on the CPU in the operator panel (OP CPU) and in teh central controller (COM CPU) provide information as to the status of the control.

When the monitor responds (LEDs light up), the axes are shut down and the PLC outputs are disconnected by the control.

During run-up of the control (POWER ON routines), both LEDs light up. In these routines it is determined which modules are in the rack and whether an operator panel is connected.

The OP CPU (operator panel CPU) and COM CPU monitor each other. If either fails to signal during the power on routines, the other remains in the STOP condition. Any interruption in the connection between the operator panel and the NC is also indicated (e.g. defective cable, OP CPU, COM CPU) by the four LEDs flashing simultaneously on the operator panel (alarm indication, motion, feed hold, program active).

LEDs remain on after powering up or come on during operation if:

- the connection between the operator panel and NC is interrupted (faulty cable etc.)
- the module jumperings are incorrect
- a module in the COM or operator panel area is defective
- the CPU is looping or was in a loop for a lengthy period, causing the monitor to trigger
- see Section 13 for additional meanings.

5.2.2 EPROM check

A cross-check sum test of the system program memory is performed with each switching on operation (in the POWER ON routines) and during operation on a cyclic basis. Any discrepancy between the specified and actual sums is indicated by the control in different ways.

The following errors are possible:

- Operator panel EPROM error
  Memory ERROR always appears in the first line on the screen after the EPROM CHECK has responded.
- Check sum error (Alarm No. 64)
  EPROM MODULE NUMBER is displayed on the screen if one or more EPROMs exhibit an incorrect check sum in an EPROM submodule. The LED of the relevant SERVO or NC CPU flashes.
- COM-CPU EPROM error
  The LED on the COM CPU flashes continuously; there is no display on the screen.

EPROM submodule number breakdown

e.g.: **570 890. 9060.11**

11 → Software version

60 → CPU code:
60 = NC
63 = SERVO
73 = PLC
00 = COM/English
01 = COM/German
02 = COM/French
03 = COM/Italian
04 = COM/Spanish
05 = COM/Dutch
06 = COM/Russian
07 = COM/Swedish
10 = COM/Finnish
11 = COM/Danish
20 = Export COM/English
21 = Export COM/German
22 = Export COM/French
23 = Export COM/Italian
24 = Export COM/Spanish
25 = Export COM/Dutch
26 = Export COM/Russian
27 = Export COM/Swedish
30 = Export COM/Finnish
31 = Export COM/Danish
81 = Operator panel board 1
82 = Operator panel board 2
83 = Operator panel board 3
84 = Operator panel board initial program loader

90 → Version (only with COM, otherwise 90):
90 = T (turning)
91 = M (milling)

890 → SINUMERIK 880 GA2

570 → SINUMERIK

5.2.3 Power-up diagnostics on operator panel

The SINUMERIK 880 GA 2 operator panel startup diagnostic display appears on the screen with each Power On.
The switch on the OP CPU must be set to 0 as standard, that is to say after the installation routine.
Switch position 8 is selected only when installing for the first time and in the event of an operator panel battery failure (see Section 6.1.1).
The same diagnostic display appears in both cases.
The following example shows the diagnostic display for an error-free startup.

System		Application	
GGS	OK	STACK	OK
VTE	OK	ROT	OK
MPC	OK	BAS	OK
POT	OK		
		DIO	OK
GMC	OK		
		DBT	OK
DEBUG	OK		
Application	OK		
Operator Panel CPU Switch	O		

The diagnostic display is automatically replaced by the basic display for the set mode after a certain time.

The messages output for the various diagnostic items can be as follows:

OK (green highlight):	No error
INIT (yellow highlight):	Data are being initialized
ERROR (red highlight):	Error or not active

Meaning of the diagnostic items:

GGS:	VDU operational.
VTE:	Data area of terminal emulation (even if Option is not available) being initialized.
MPC:	Connection established between operator panel and central controller.
POT:	Keyboard processing and I/O submodule handling are OK.
GMC:	VDU operational.
DEBUG:	This system message indicates that the RMOS debugger is ready (relevant only for development purposes).
Application:	RMOS operating system has been initiated and is now starting the application.
Stack:	The application stack is tested at switch position 0 and initialized at switch position 8.
ROT:	The startup routine for the application is initiated.

BAS: The operation sequence control for the operator panel is initiated.

DIO: RS232C (V.24) interfaces of the operator panel are ready.

DBT: The data link via the general data channel between the operator panel and control is operational.

The "Operator Panel CPU Switch" display indicates the position of the rotary switch on the operator panel CPU.

Meanings relevant to the user:

The following system messages are of significance to the user and must be acted on accordingly.

STACK ERROR

If this error occurs, the operator panel installation routine has to be executed (Power On with operator panel CPU rotary switch set to "8").

MPC INIT

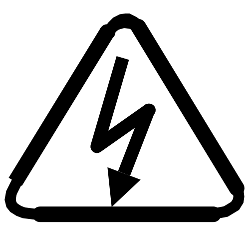
The system remains in this constellation with Power On. The operator panel cannot by linked to the control via the MPC.
If this error occurs, the control has to be powered up in the startup mode (of the COM CPU).

POT INIT

The system remains in this constellation with Power On. The link between the OPI (Operator Panel Interface) board and the INTKEY (Interface Keyboard) board cannot be established.
If this error occurs, press the reset button for the INTKEY board on the operator panel and initiate a control Power On.

5.2.4 Brightness adjustment

Die SINUMERIK 880 GA2 is delivered with a 12” colour monitor.



Caution!
High voltage approx. 16 kV in visual display unit and at high-voltage transformer, anode lead and anode terminal at CRT.

The brightness of the colour monitor is adjusted by means of a potentiometer on the back of the housing.

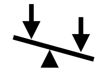
Correct adjustment of the picture hold, height, width, contrast etc. is usually performed at the factory. Alterations to the potentiometer settings on the monitor PCB should only be performed by trained personnel using special tools (epoxy resin screwdrivers).

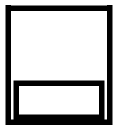
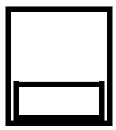
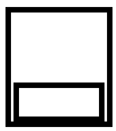
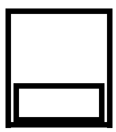
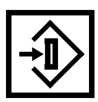
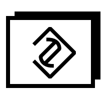
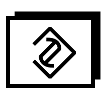
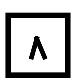
The guidelines issued by the Federal German Employers' Liability Insurance Association for Precision Mechanics and Electrical Engineering (Berufsgenossenschaft der Feinmechanik und Elektrotechnik) should be observed.

5.3 Drift compensation

On SINUMERIK 880 GA2 drift compensation is semi-automatic, because the operator must activate it. If the axes (of the CNC) and drives are in closed-loop control, drift compensation may be performed.

The drift has exceeded the permissible value specified in NC MD 204* and 208* if the green LED fails to go out after travel (axis stopped). Drift compensation must be performed.



	DIAG- NOSIS		Press softkey 'DIAGNOSTICS'
	MACH. DATA		Press softkey 'MACH. DATA'
	NC MD		Press softkey 'NC MD'
	AXIAL DATA 1		Press softkey 'AXIAL DATA 1'
	2720		Enter numerical value 2720 (1st axis) and actuate SEARCH key
or	2721		With the EDIT key the following error is calculated separately for each axis
:	:		
or	2731		The EDIT key must be pressed for every axis
			Actuate RECALL key (return) three times

Drift compensation must be performed individually for all axes.

Drift compensation may also be performed manually by changing the value in MD 272* until the following error at rest is zero (check the service data for the axes).

Caution!

With very precise machines, drift compensation should be performed several times daily on account of the temperature differences during operation since drift directly affects the following error.

An "automatic drift compensation" can be programmed on SINUMERIK 880 GA2.

With both @361 (axis actual value machine related) and @362 (axis actual value machine related incl. following error) the drift can be calculated as follows from the following error and entered in NC MD 272* in a subroutine:

R1 = axis number
R0 = following error/drift

Example:

`@361 R0 R1`	Read axis actual value machine related
`@362 R2 R1`	Read axis actual value machine related incl. following error
`R0 = R0 - R2`	The following error is in R0
	Conversion of the following error including the servo gain factor (NC MD 252*) and the multgain (NC MD 260*) in VELO. Enter the result additively in NC MD 272* (drift).
`:`	
`R4 = R1 + 2720`	
`@400 R4 R0`	Check whether following error is ZERO.

Restrictions:

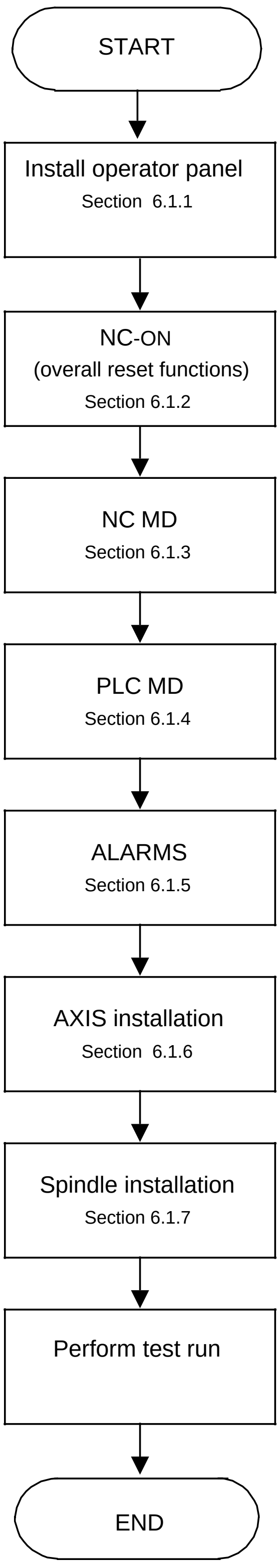
- For linear axes (more complicated for rotary axis)
- When the subroutine is called (@361+@362 to NC MD 272*) the axis in question must be stopped.

6 Standard Installation Sequence

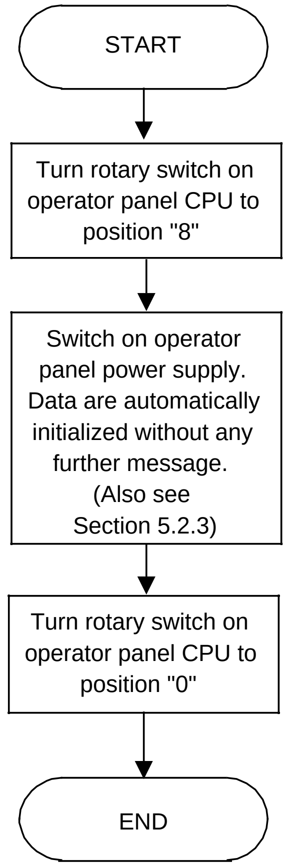
Comply with the following sequence for standard installation of the SINUMERIK 880 GA2:

1. Connect the control and external components in accordance with the 880 GA2 operating instructions (Instruction Manual), the Interface Description Part 2 and the universal interface description.
2. Check the cables and connections with reference to the Interface Description, Part 2.
3. Note Section 1 of the Installation Guide.
4. Axes and spindles, also speed controller, already installed.
5. Hardware for servo disable connected for all axes and spindles.
6. All PLC input/output modules and handwheels connected.
7. Perform installation routines in Section 9.2.1.
8. Note installation of operator panel and control according to Section 6.1.
9. Check all data, expecially NC MD, PLC MD and setting data, for permissible values (see Section 9).
10. Note Section 2 of the Installation Guide and append the Checklist to the completed log book.

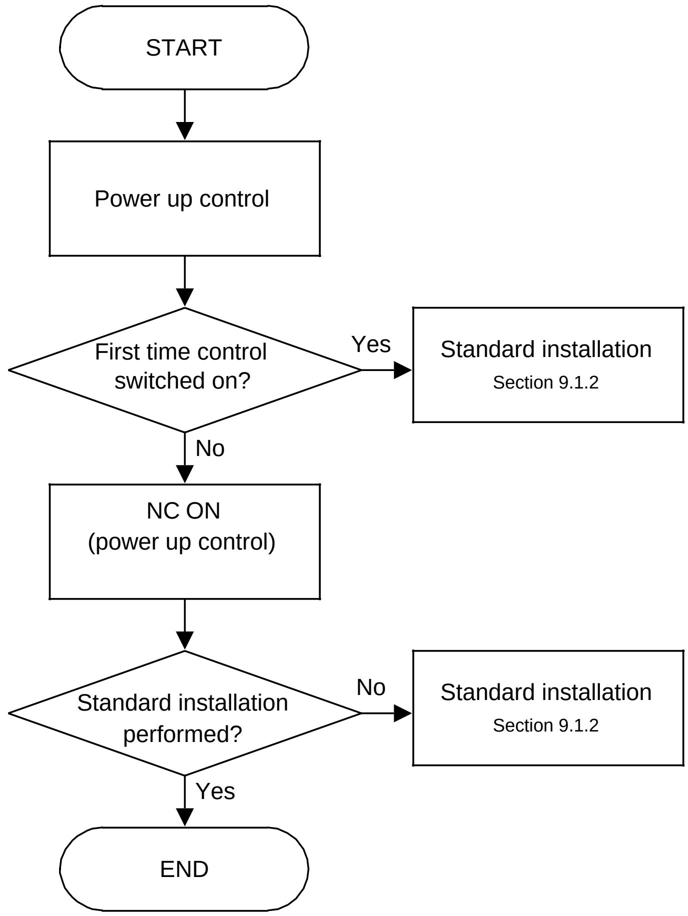
6.1 Standard installation shown as a flowchart



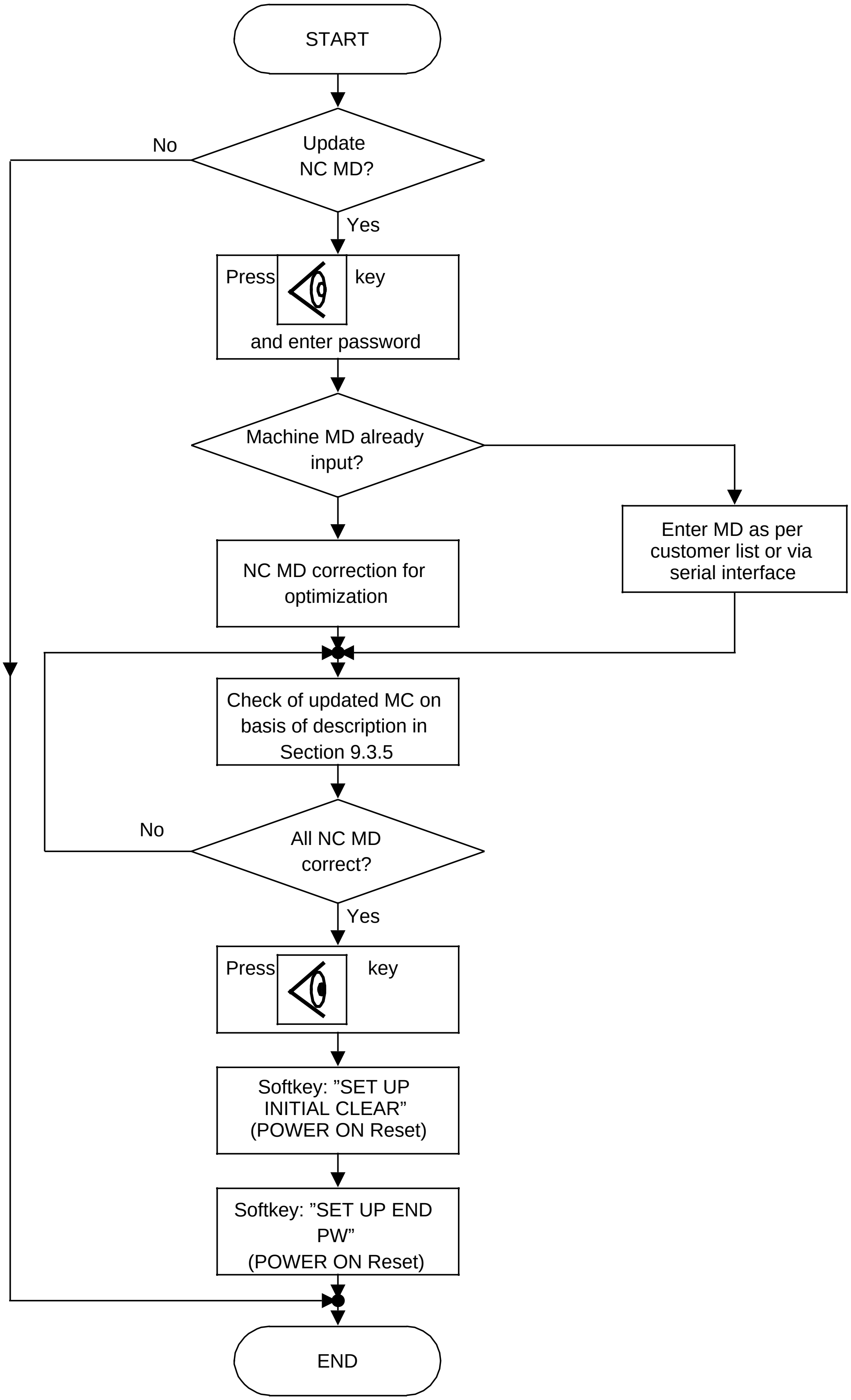
6.1.1 Operator panel installation



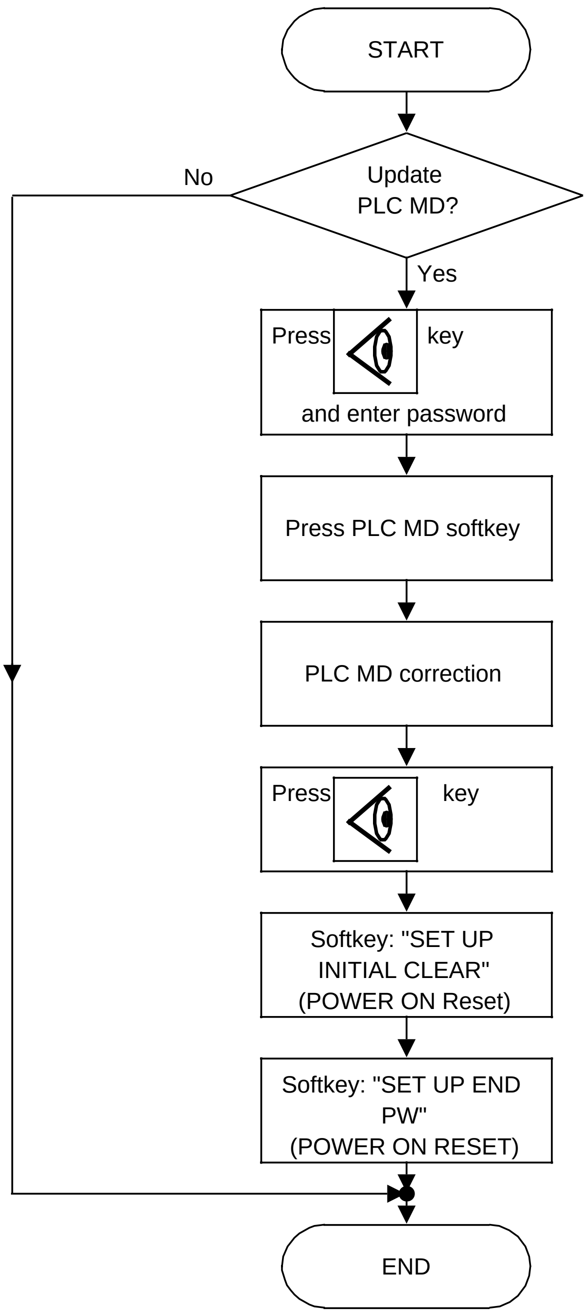
6.1.2 NC ON (overall reset chart and setting standard NC MD)



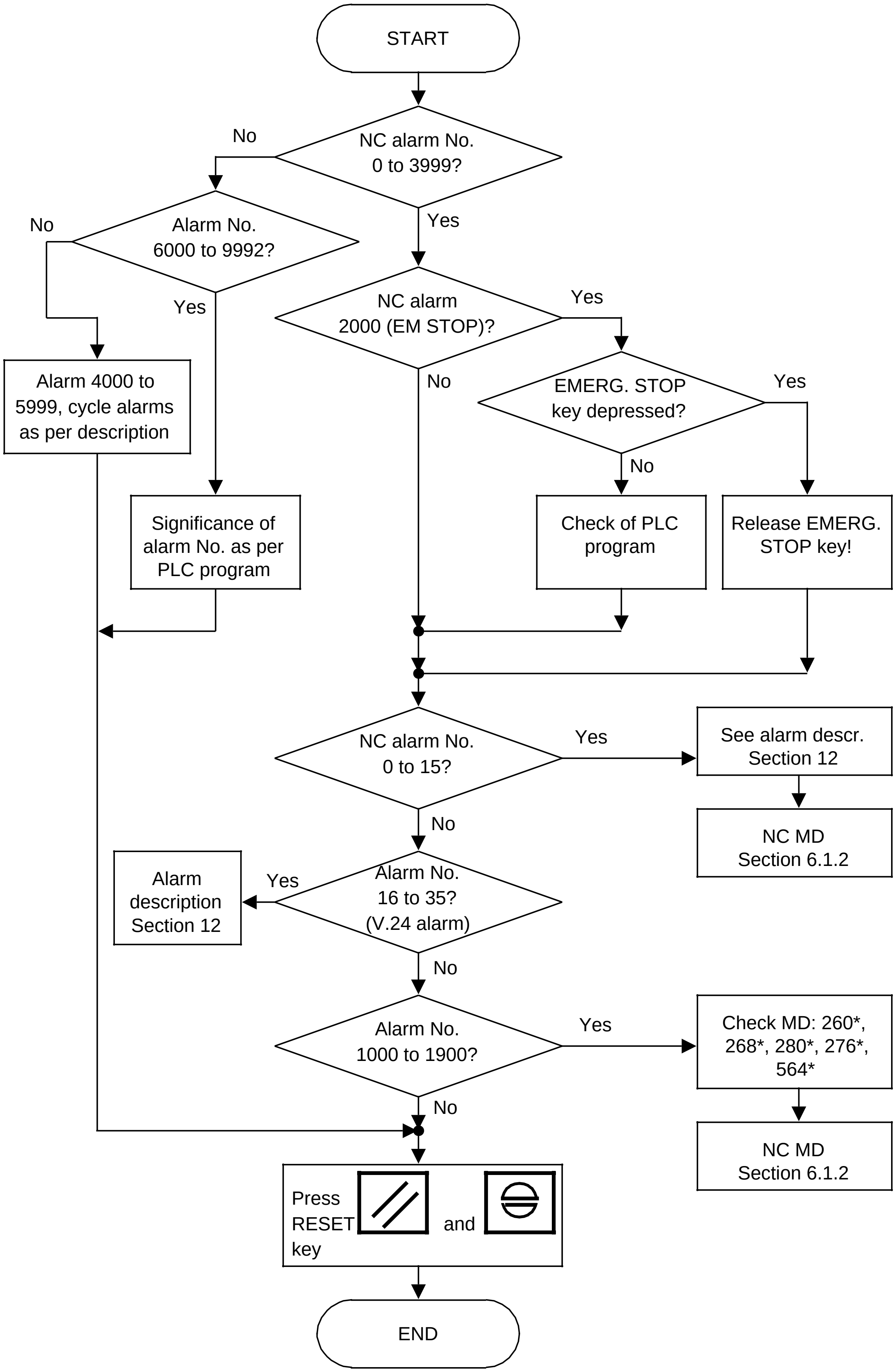
6.1.3 NC machine data



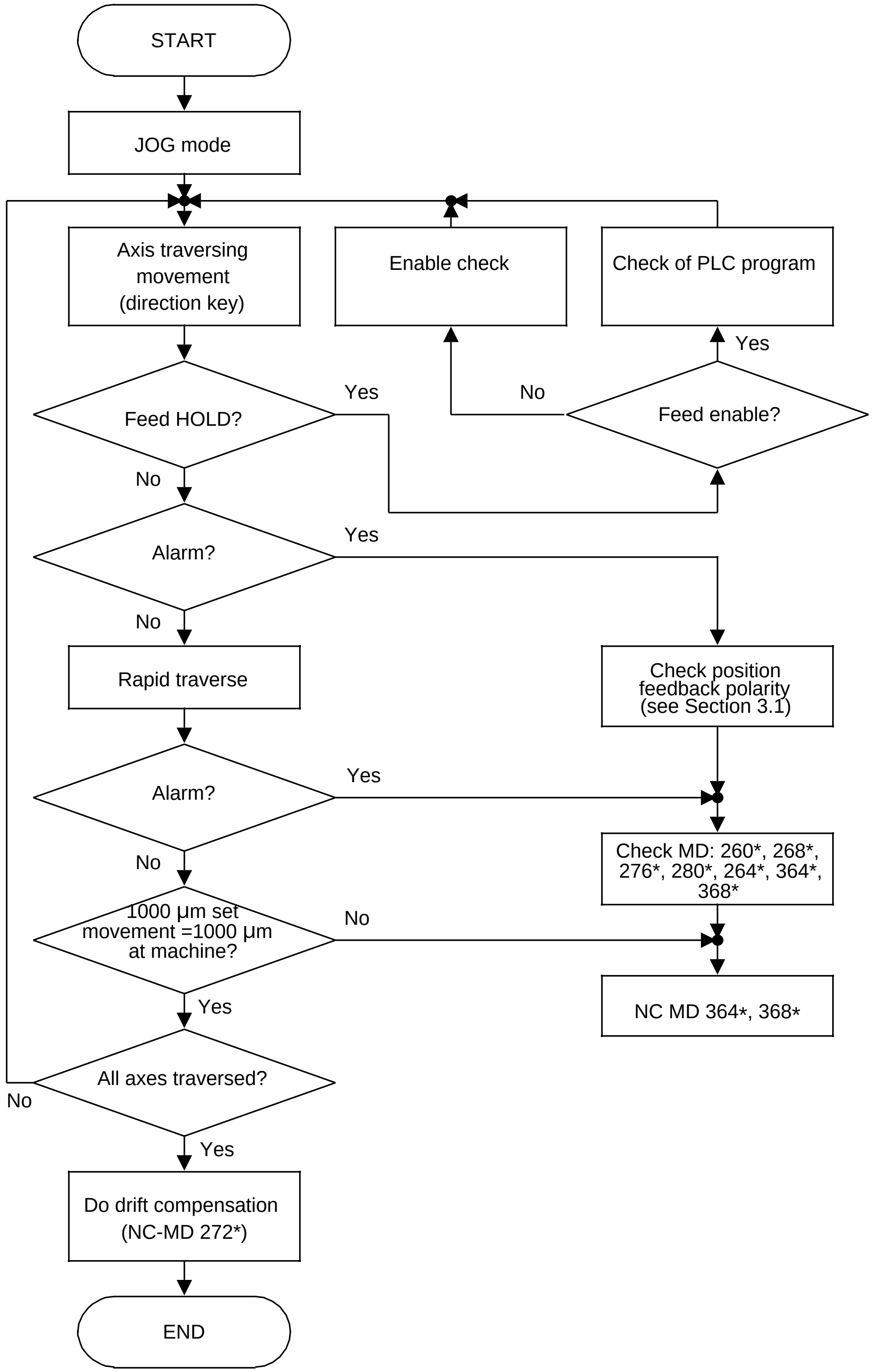
6.1.4 PLC machine data



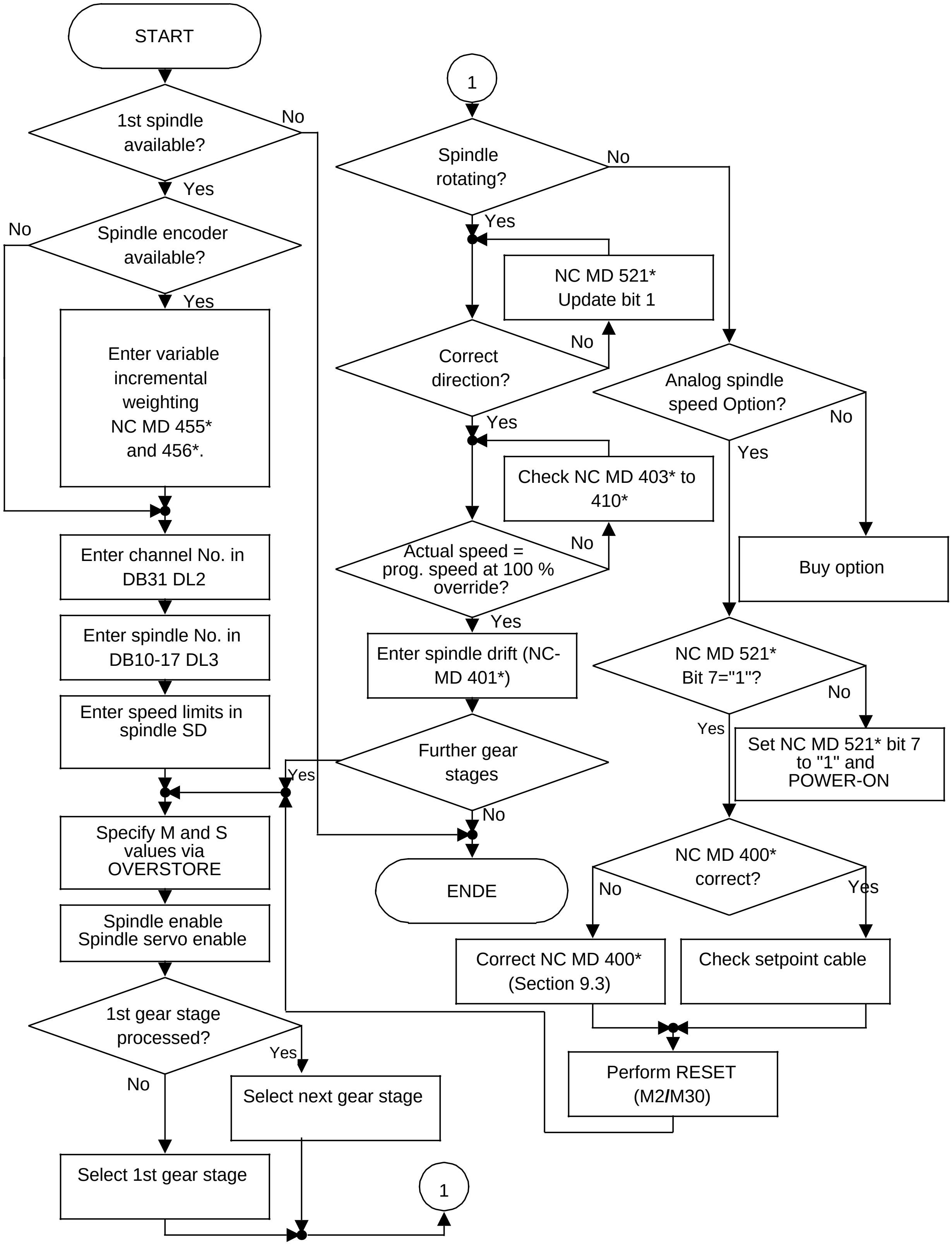
6.1.5 Alarm processing



6.1.6 Axis installation (simplified flowchart)



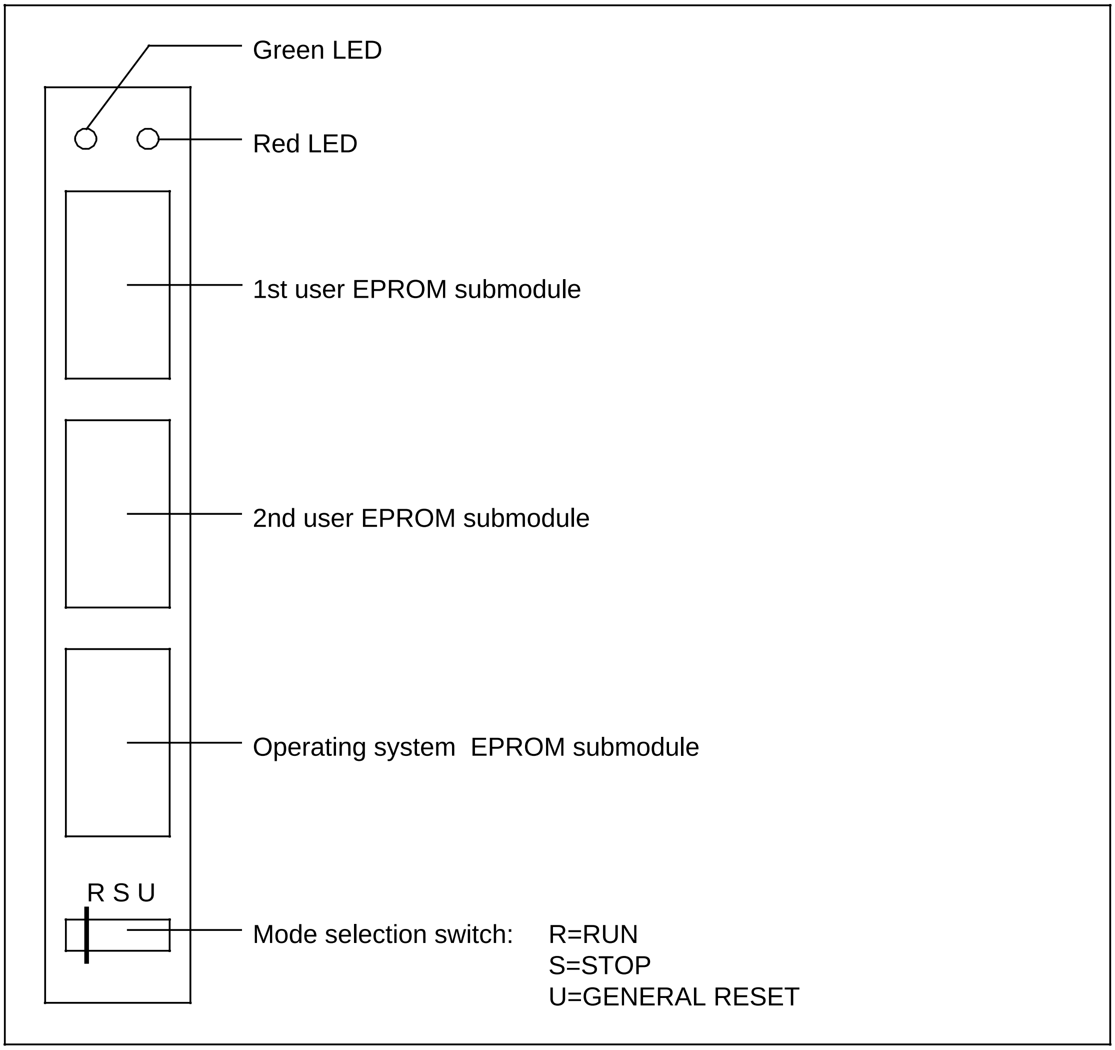
6.1.7 Spindle installation



7 135 WB PLC Description

7.1 General

On SINUMERIK 880 GA2 the PLC CPU 135 WB is used. This PLC runs as a ”RAM machine”, i.e. the operating system and the user program are loaded from the EPROM into internal RAM on a general reset. This makes processing considerably faster.



Front view of 135 WB

Its performance can be expanded, depending on the control frame configuration, by connecting up to a total of 2 PLC CPUs. The technical data and notes on programming are contained in the Planning Guide (Configuring Instructions) entitled "Configuring the 135 WB PLC".

7.2 Installation

Installation of the PLC 135WB interface control is subordinate to the installation of the numerical machine tool control (NC). The CPU of the PLC has subordinate functions in this case.

When the NC branches into the "Setting-up mode initial clear" mode a setting up bit is set. In the NC, deletion and loading of the NC and PLC machine data for parameter assignment of the system program can be executed, with the aid of which setting up of the PLC 135 WB is controlled.

7.2.1 Self-diagnotics program

After switching on the mains voltage, the interface control runs a self-diagnostics program. This program tests the most important hardware components and initializes the software required for system start up.

If errors in the system are recognized, the LED on the front plate displays the error.

LED	Significance
Continuous (green only)	Cyclic operation
Continuous (red only)	Stop state
Continuous (red and green)	INITIAL CLEAR required (initial power on or data loss)
Flashing (red) once	Error with cross-check sum via the system program
twice	Error in the CPU RAM test
3 times	Error Timer 0 (process-internal timer) or Watchdog error
4 times	Error in monitoring test for timeout
5 times	Acces to link RAM not possible. This can also happen, for example, if a power failure takes place while a large file is being copied etc.
6 times	Error with test access to link RAM
7 times	Error in system initialization program
8 times	Other PLC failure (with DUO PLC)
9 times	SINUMERIK 840: synchronisation error in PG link
10 times	Error of the internal coprocessor (COP) register or in the step address counter (SAZ) creation
11 times	Command delegation to word processor (WOP)
12 times	Processing of binary commands
13 times	Processing of OR, bracket expressions, NOP or BLD commands
14 times	Processing of block calls and jump commands
15 times	Processing of timer and counter operations
16 times	Addressing in data memory
17 times	Command execution mode
18 times	Test address comparator machine code (MC5) and interrupt processing via coprocessor
19 times	Monitoring test for timeout with access of the coprocessor to the user memory
20 times	ACOP-2 not plugged in

7.2.2 System initialization program

After the self-diagnostics program has been run through, the system initialization program is requested.

In its first section, the data required for running the organization program are set up. This setting up includes:

- Stack organization,
- Segmentation for word processor and coprocessor,
- Entries in the location-dependent CPU interrupt table,
- Task priority lists,
- Setting up task data,
- Initialization of counts and periodic values.

In the second section the system initialization program defines the type of start up after switching on the mains voltage. The following points are checked:

- Whether the switch-on test pattern is missing (i.e. data lost)
- Whether there is a battery interrupt
- If the setting-up bit is set
- Request from the NC "automatic warm restart after setting-up initial clear"
- STOZUS operating status bit set (accquisition of interrupt event or continuation of the STOP state, see Section 8)
- Cold restart or warm restart attempt aborted.

If the STOZUS identifier is set, the control remains in the STOP state.

If, in the second section, (testing of run-up after switching on the mains voltage) the STOZUS identifier is not set, but one of the other conditions is fulfilled, an automatic cold restart is executed; a warm restart of the Control only occurs if none of the mentioned conditions are fulfilled.

Initial clear with subsequent bootstrapping of the user memory (URLOE = 1) is always required.

- If first setting up has been carried out,
- Data loss has occurred by removing the PLC CPU or, in the case of power failure, due to simultaneous battery voltage failure.

If the mains voltage fails during active processing checks, the processing checks are aborted by the programmer. The system initialization program causes a cold restart.

7.2.3 Timeout analysis

A write access to the communication or local bus is executed by the bus interface. The processor immediately receives an acknowledgement and continues. (Buffered access to communication/local bus). If a timeout occurs during such an access, the current state of the registers of the processor and coprocessor give no information as to the cause of the timeout.

The user can switch off buffered accesses to the communication and local bus (e.g. to test STEP 5 programs during the installation phase) via machine data (PLC operating system MD bits 6049.0). These accesses are then slower because the processor only receives an acknowledgement when the whole bus cycle has finished.

Machine data 6049.0 must be set in order to be able to determine the exact cause of a timeout.

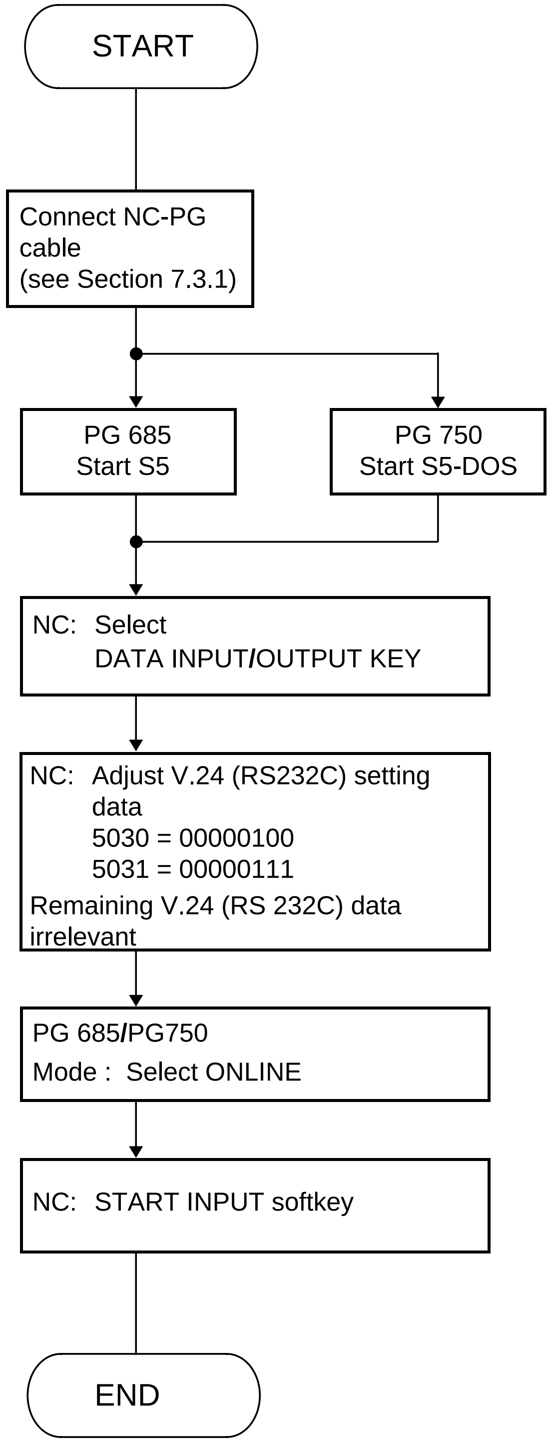
7.2.4 Error list for PLC 135 WB

The operating system can detect malfunctioning of the central processor, errors in the system program or the effects of erroneous programming on the part of the user.

If, while executing a command, the interpreter encounters another error which will result in a program interrupt, it branches to the STOP loop.

A more exact error analysis can be carried out with the help of the PG 685 or 750 or the detailed error code which is integrated into the control. The interrupt stack at the programmer and/or the NC in the PLC status display in DB 1 in DW 160 to 164 and the detailed error code, which are all described in Section 8 of Installation Lists, are available for this purpose.

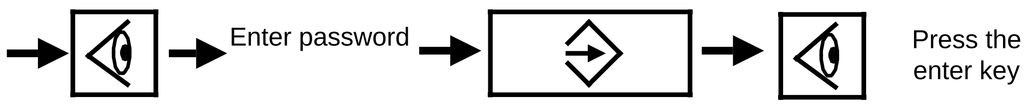
7.2.5 PLC-PG 685/PG 750 link



7.3 Installation of the 135 WB PLC

During installation the 135WB PLC is in the stop condition (red and green LED).
For installation purposes, the 135 WB PLC is subordinated to the COM CPU.
For initial start-up, the standard NC machine data are loaded first and then, depending on the number of PLC CPUs, NC MD 5038 bits 0 and 1 are set.

Operator input:



INITIAL CLEAR	SET UP END	SET UP END PW		

The control is in the "INITIAL CLEAR" mode.
The PLC MD then have to be erased and the standard machine data loaded.

Operator input:

DATA IN-OUT	NC DATA	PLC UTILITY	**MACHINE DATA**	SET UP END PW

CLEAR NC MD	LOAD NC MD	**CLEAR PLC MD**	LOAD PLC MD	CLEAR CYC. MD

Clear PLC MD

CLEAR NC MD	LOAD NC MD	CLEAR PLC MD	**LOAD PLC MD**	CLEAR CYC. MD

Load PLC MD

The standard machine data are loaded.

The "INITIAL CLEAR" function is executed next.

Operator input:

DATA IN-OUT	NC DATA	**PLC UTILITY**	MACHINE DATA	SET UP END PW

PLC INITIAL				

PLC1 INITIAL	PLC2 INITIAL			

				SET UP END PW

7.4 PLC DIAGNOSTICS

In the PLC diagnostics mode the user or operator is able to view the error/status messages (referred to the channel/spindle /axes) and the event group displays (referred to PLC1/PLC2). The sense bits that obtain the message displays are set by the PLC program (also see Interface Description, Part 1, Signals, Section 19).

In all operating modes the "DIAGNOSTICS" softkey is in the first row (with the exception of "PRESET" mode where the "DIAGNOSTICS" softkey is not displayed at all).

Operator input:

DIAGNOSTICS → ALARM MESSAGE → PLC DIAGNOSTICS

These softkey functions should be selected one after the other

CHANNEL	SPINDLE	AXIS	PLC1	PLC2

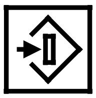
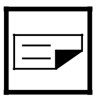
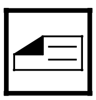
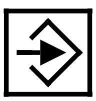
This softkey menu is displayed

7.5 PLC STATUS

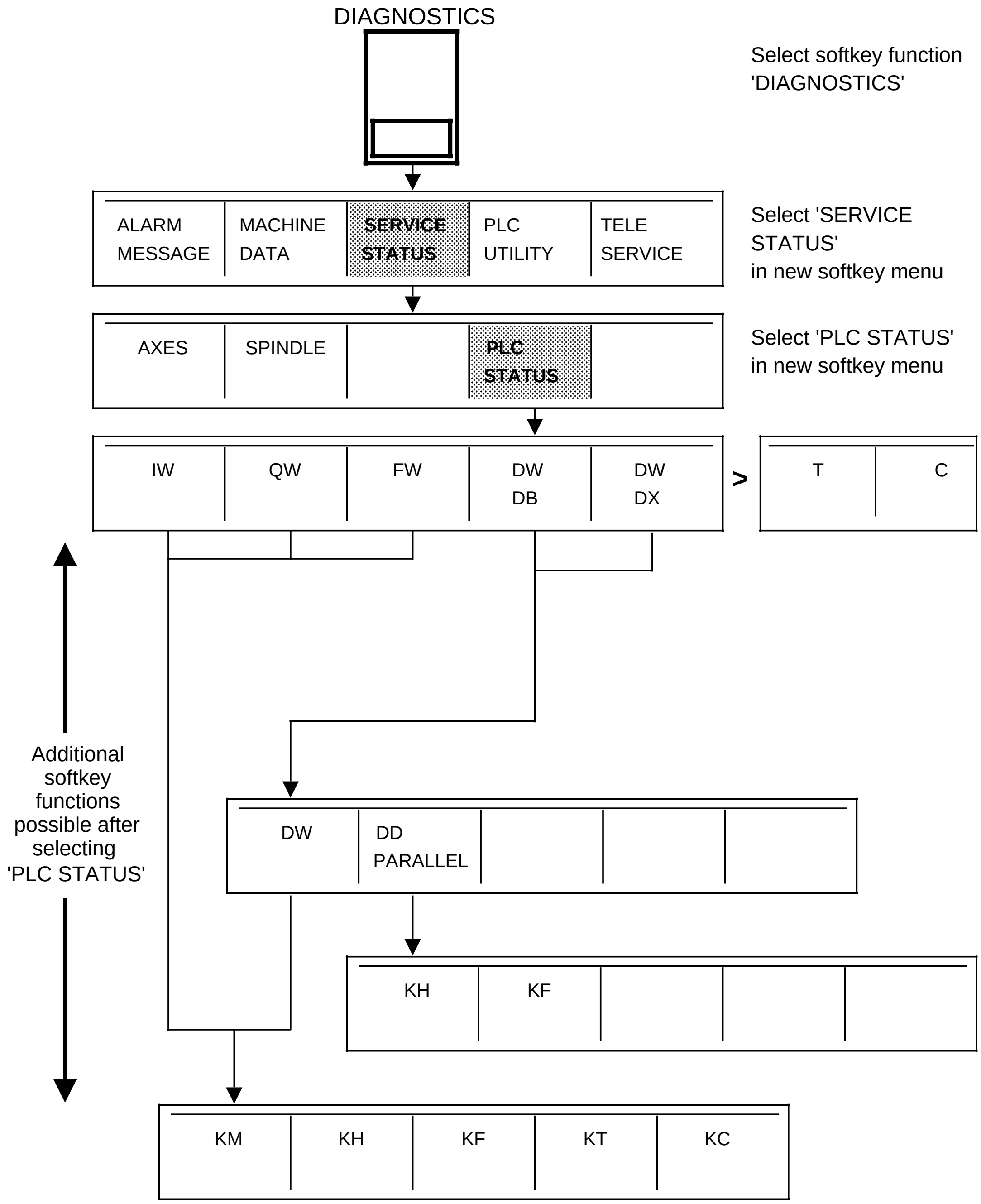
In ”PLC STATUS” mode, counters and timers can be read, and input words, output words, flag words, data words and double data words can be read and written. These words may only be written following an enabling operation as a result of entering the password.

Starting with the preselected No., consecutive input, output, flag and data words are shown in the PLC status display (not with timers and counters).

7.5.1 Operation of PLC STATUS

Key		PRESELECTION:	Each existing byte number can be preselected
Key		Page DOWN:	Byte number incremented by one
Key		Page UP:	Byte number decremented by one
Key		INPUT:	Change value in selected word or bit number by re-entering
Key	∧	RECALL:	Return to previous display
Key	>	EXTEND SOFTKEY MENU:	Extend called display

7.5.2 PLC STATUS selection



Select softkey function 'DIAGNOSTICS'

Select 'SERVICE STATUS' in new softkey menu

Select 'PLC STATUS' in new softkey menu

7.6 Use of distributed I/O devices

The SINUMERIK 880 GA 2 envisages two types of link for the operation of distributed I/O devices.

1. DMP link via Interface DMP
2. Parallel link via INT EU/16 bit for interfacing SIMATIC EU 185U

A maximum of three Interface DMP modules or one INT EU/16 bit interface plus max. one Interface DMP module are permitted per PLC.

If different links are used on the PLC, the firmware executes a prioritization routine. The following are recorded in turn:

1. Interface type INT EU/16 bit (max. one allowed per PLC), always treated as the 2nd interface
2. All Interface DMP modules according to the sequence of their slots

Example: INT EU/16 bit in slot n
Interface DMP in slot n + 1

Therefore:	INT EU/16 bit	2nd interface
	Interface DMP (slot n)	1st interface

The automatic numbering as described above is directly related to

- the associated machine data required for configuring the interfaces
- the configuration data block in the case of free configuration.

The address decoding of the interface in the address space of the PLC also takes place according to this numbering. A change in configuration can therefore also alter the location of the interfaces' address space. This is of significance, for example, when using LIR/TIR commands.

Note: Section 7.7 contains a more detailed description of "free configuration".

The stated machine data and configuration DBs must be set according to the number of the interface.

In the example described above the PLC machine data of the 1st interface (n + 34 to n + 63) would be responsible for the Interface DMP at slot n + 1 in the case of free configuration. The INT EU/16 bit would be the 2nd interface, but does not need any PLC machine data.
(n = 0 for 1st PLC, n = 200 for 2nd PLC)

I/O device faults

The PLC generally enters the STOP state in the event of I/O device malfunctions. The failed I/O modules are reported to the user in DB1. PLC MD can no longer be used to stipulate whether the PLC is to enter the STOP state, as was the case with SINUMERIK 880 GA 1.

Startup behaviour

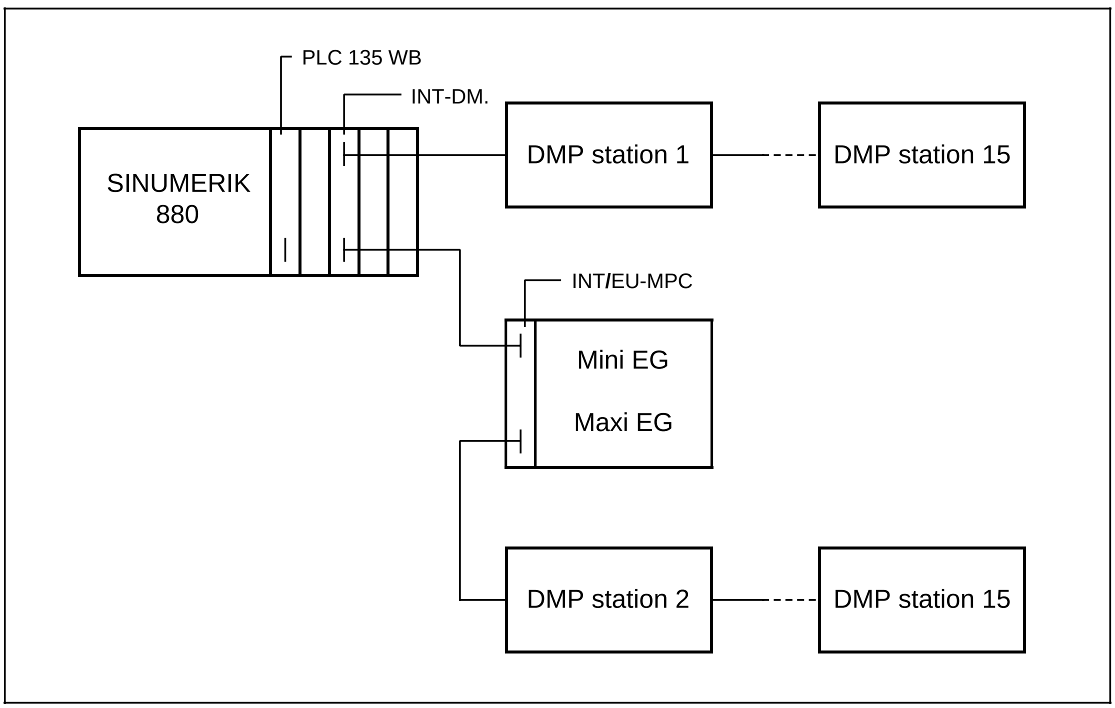
If the interfaced I/O devices are still inoperable (power supplies not yet connected, connection not yet made, or if the submodule is declared but incorrectly jumpered at the rotary switch), the relevant startup method is interrupted and the PLC branches to the STOP state. It remains in this state until all the conditions for a startup are fulfilled (I/O devices operable). Once this is the case, a common system runup is executed on PLC initiative, whereby the PLC runs up by the appropriate method (cold or warm restart). As soon as the PLC has assumed the STOP state with inoperable I/O devices, a message is issued for the user in the detailed error code (see Installation Lists).

7.6.1 Interface DMP

The Interface DMP module has two separate MPC interfaces (lines), each of which leads to two front panel connectors.

The following I/O devices can be connected to each MPC interface:

DMP 16 inputs/outputs
DMP 32 inputs
Compact DMP module
IP65 DMP module
Mini EU: Connected via INT CU/MPC in EU
Maxi EU: Connected via INT CU/MPC in EU



Here again, the maximum number of modules that can be connected per line (MPC interface) is 15.

Instead of DMP stations, up to three expansion units can be operated in each line with the Interface DMP. In this case the number of stations is reduced by the number of connected EUs.

Both the SINUMERIK standard I/O devices and the released SIMATIC I/O devices can be used in the expansion units.

Machine data

m+24 – m+48	:	Initial addresses for DMP stations at 1st DMP interface, 1st MPC interface (line)
m+49 – m+63	:	Initial addresses for DMP stations at 1st DMP interface, 2nd MPC interface (line)
m+64 – m+78	:	Initial addresses for DMP stations at 2nd DMP interface, 1st MPC interface (line)
m+79 – m+93	:	Initial addresses for DMP stations at 2nd DMP interface, 2nd MPC interface (line)
m+94 – m+78	:	Initial addresses for DMP stations at 3rd DMP interface, 1st MPC interface (line)
m+109 – m+123	:	Initial addresses for DMP stations at 3rd DMP interface, 2nd MPC interface (line)

PLC 1: m = 0, PLC 2: m = 200

The following relationship exists between the station declaration in the machine data, the device No. in the configuration DB, and the rotary switch on the station:

MD designation	Device No. in configuration DB	Rotary switch
1st DMP station	1	E
2nd DMP station	2	D
3rd DMP station	3	C
4th DMP station	4	B
5th DMP station	5	A
6th DMP station	6	9
7th DMP station	7	8
8th DMP station	8	7
9th DMP station	9	6
10th DMP station	10	5
11th DMP station	11	4
12th DMP station	12	3
13th DMP station	13	2
14th DMP station	14	1
15th DMP station	15	0

If expansion units are used in the line, they must be jumpered to 0 - 2 at the INT CU/MPC rotary switch.

For the purposes of recording the expansion units, assign the machine data (initial addresses) stated above or data words in the project block (n + 4, n + 5) a value other than the standard value (-1). The Interface DMP interprets the relevant MD or DW in the case of an EU only as an "available declaration"; it independently checks the complement in the EU and reports this to the PLC. Here again, the same applies accordingly to modules; only declared EUs are recorded, operated and monitored.

NC MD	130 – 135:	Interrupt bytes of DMP interfaces
NC MD	6068 – 6073:	Edge selection of interrupt inputs
NC MD	6074 – 6079:	Enable of interrupt inputs

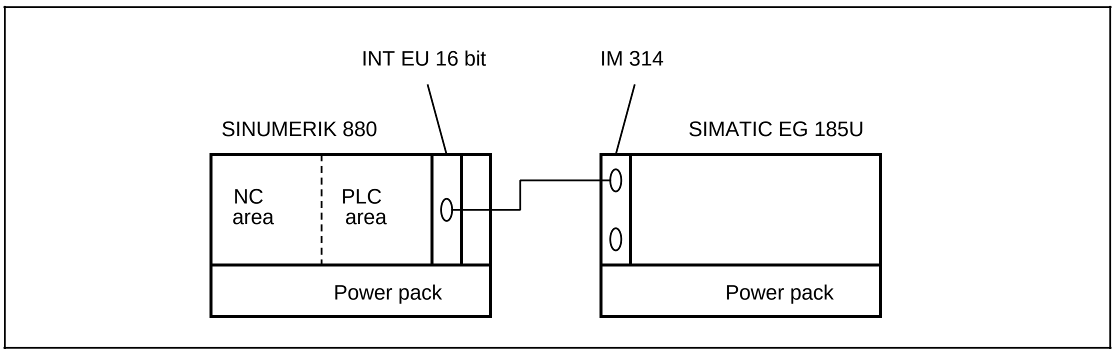
The interrupt input signal changes selected via NC MD are recorded in FYs 8 to 11.

See the Interface Description Part 1, Signals, Section 5.3 for notes on FYs 8 to 11.

7.6.2 INT EU/16 bit

The SIMATIC expansion unit EU 185U (6ES5185-3UA...) is connected to the PLC 135WB by a parallel link.

For this purpose the INT EU/16 bit module (6FX1137-8BB02) is used as the expansion unit interface and module IM 314 as the central controller interface in the EU 185U.



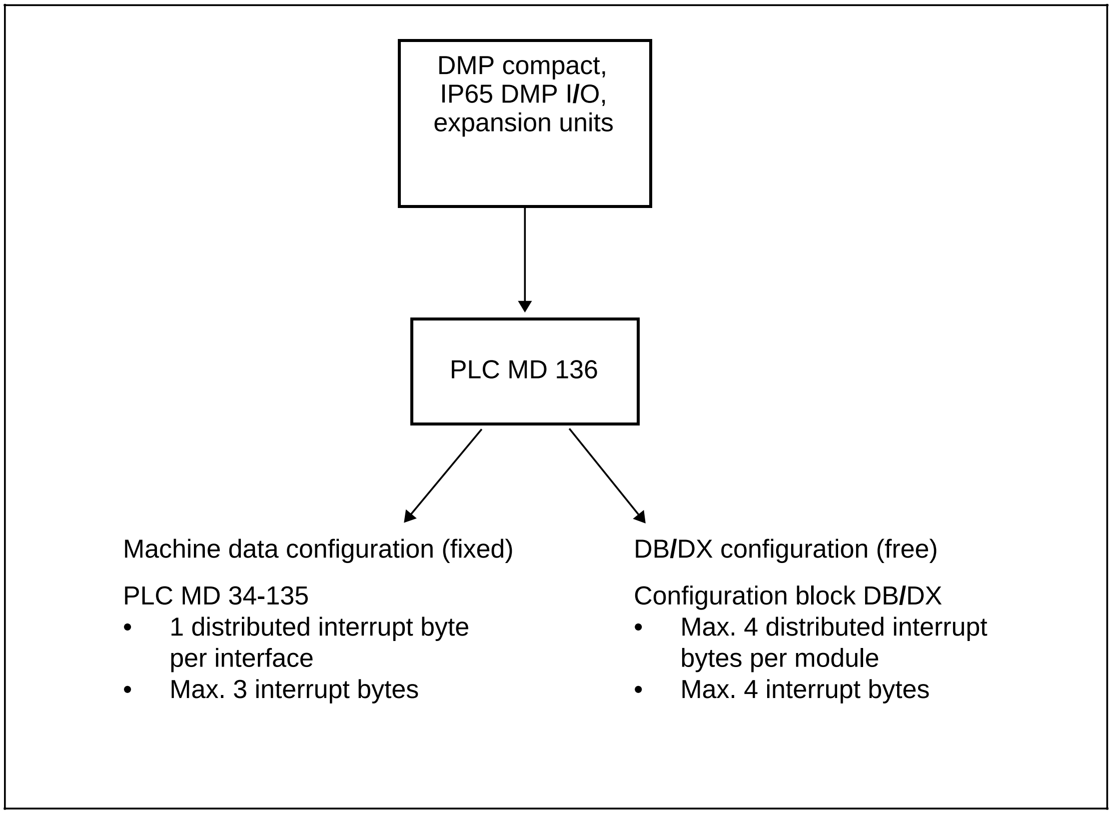
Only one EU interface, which can be operated at any slot of the PLC bus segment, can be inserted per PLC.

The address decoding of the SIMATIC I/O submodules used in the EU takes place on the modules themselves (Dip Fix switches).

Machine data

No machine data are required for declaring the interfaces or connected expansion units. Similarly, no declaration is required in the case of free configuration either.

7.7 Configuring distributed I/O devices



The distributed machine peripherals can be addressed either by machine data or by configuration blocks. A mixed configuration is not allowed.

7.7.1 Configuration with PLC machine data (fixed configuration)

The initial addresses of the DMP station input and output bytes used are specified with PLC MDs 34 to 123. The PLC MD assigns the same initial address to both the input and output bytes.

Example: DMP module 16 inputs/16 outputs

PLC MD 34 = 74

1st IB = IB74, 2nd IB = IB75
1st QB = QB74, 2nd QB = QB75

Example: DMP compact station

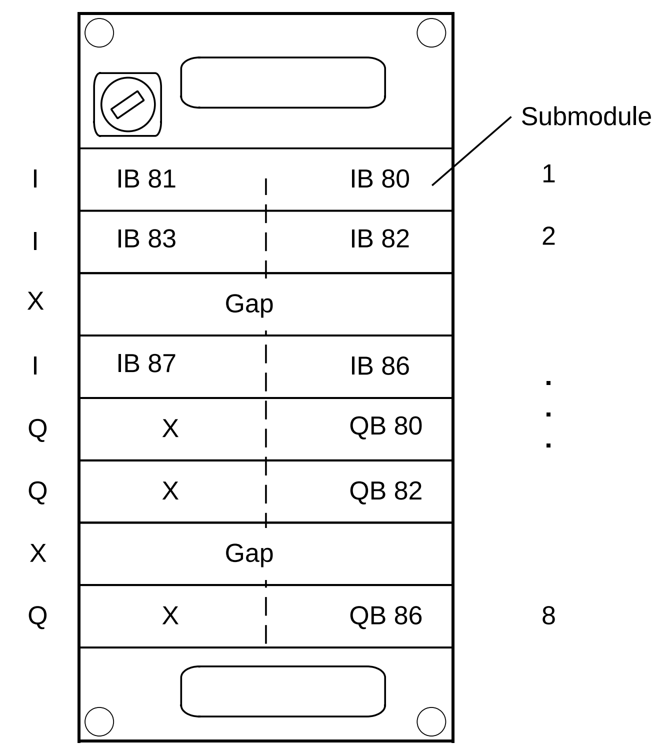
PLC MD 34 = 80

The initial addresses of the input and output bytes are again specified by a PLC MD. The input and output module slots are addressed consecutively starting from this byte address.

With the DMP compact terminal block a maximum of 4 input submodules (16 inputs) and 4 output submodules (8 outputs) can be inserted.

The modules are word addressed. In the case of the output modules, however, only one byte can be assigned.

A fixed slot assignment exists for the input/output modules.



DMP compact station

Interrupt routing

One interrupt byte is permitted per interface (note: not per line):

PLC MD 130 No. of interrupt byte of 1st interface 1st line

or

PLC MD 131 No. of interrupt byte of 1st interface 2nd line

⋮

PLC MD 134 No. of interrupt byte of 3rd interface 1st line

PLC MD 6068 ... 6072 Interrupt edges
PLC MD 6074 ... 6078 Interrupt edges.

7.7.2 Free configuration with configuration DB/DX

Free configuration allows separate setting of the initial addresses for the input and output bytes of the DMP stations as well as of the individual submodules of the DMP compact station.

The switchover between conventional configuration (via PLC MD) and free configuration (with the aid of a configuration DB or DX) is made by

PLC machine data 136 : Determining the method of configuration for DMP stations

0 : Standard value, conventional configuration, i.e. PLC MDs 34 to 123 and 130 to 135 and PLC MD bits 6068 to 6079 are valid.

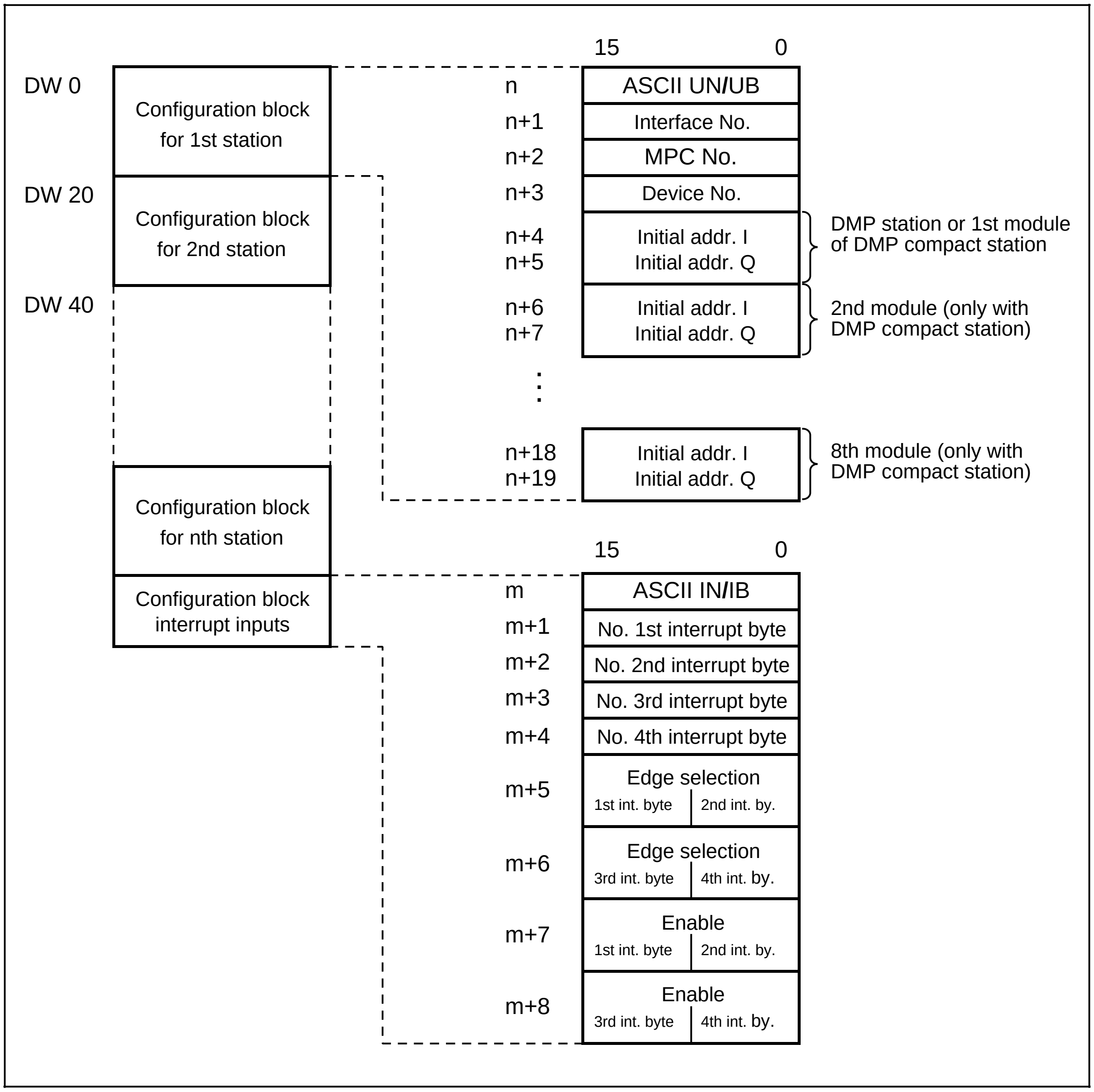
0 : The number of the DB used for the configuration data is entered by the user. This specifies the extended addressing of the I/O submodules (DMP compact); the PLC MDs stated above are not evaluated.

DX blocks can be used:

DB 1 ... 255	Entry	1 ... 255
DX 0 ... 255	Entry	1000 ... 1255

Free configuration is especially suitable for DMP stations (including DMP compact). It also allows modules with different addressing principles, e.g. SINUMERIK EUs, to be assimilated. In such cases, the entry for the 1st module is obligatory as the initial address of the entire block or as the identifier for an available EU.

The configuration DBs are structured as follows:



A configuration block for a DMP station has a fixed length of 20 data words; an interrupt block is 9 data words long.

Configuration block for DMP station

- ASCII UN/UB: Block identification with ASCII characters

  UN (format KC); 554E (format KH) = Configuration block is available and its values are effective. The relevant DMP machine data are ineffective.

  UB (format KC); 5542 (format KH) = Configuration block is available but declared invalid. The entered values are ineffective.

- Interface No.: Number of the Interface DMP module (also see Section 7.6)

  1 = 1st Interface DMP module
  2 = 2nd Interface DMP module
  3 = 3rd Interface DMP module

- Device No.: Number of the DMP terminal block in the line (1 to 15), set with the rotary switch

- Initial address I/Q: Initial byte number of the inputs and outputs on the DMP station. Addressing is always word by word so that, in the case of a DNP with 8 outputs, the system reserves the addressed byte and the following one although the following byte is not used.

Configuration block for interrupt bytes

- ASCII IN/IB:Interrupt identification with ASCII characters

  IN (format KC); 494E (format KH) = Interrupt block is available and its values are effective. The relevant DMP machine data are ineffective.
  IB (format KC); 4942 (format KH) = Interrupt block is available but declared invalid. The entered values are ineffective.

- Interrupt bytes 1 to 4: Declaration of the 4 DMP interrupt input bytes (value range 0 to 254). A DMP station can also accommodate several interrupt bytes.

- Flank selection: Bit-by-bit assignment of the interrupt triggering edges (1 DW for 2 INT bytes)

  0 = 0 1 edge
  1 = 1 0 edge

- Enable: Bit-by-bit enable of the interrupt inputs (1 DW for 2 INT bytes)

  0 = Interrupt input disabled
  1 = Interrupt input enabled

If the configuration DB is missing (in MD 136) or is faulty, i.e. contains illegal values or if data words are missing (incomplete block lengths), the PLC enters the STOP state with error message 33 and additional information on the error.

Interrupt routing

- A total of 4 interrupt bytes are permitted.
- A maximum of 4 interrupt bytes can be used per module and per interface.
- The interrupt triggering peripheral bytes can be distributed between the modules if desired.
- All the interrupt bytes of a line must be on the same module.
- No interrupt bytes can be declared in a line with an EU.
- OB 2 must be enabled. PLC MD 6050.2 and the PLC CPU must be operating in the special mode. PLC MD 6051.0 = 0.
- OB 2 is activated with each interrupt request. In addition, the interrupt source is indicated in flag bytes FY 8 to FY 11.
- Flag bytes FY 8 to FY 11 must be reset by the user in OB 2.

Definition of flag bytes FY 8 to FY 11

FY 8	Interrupt inputs 1st INT byte
FY 9	Interrupt inputs 2nd INT byte
FY 10	Interrupt inputs 3rd INT byte
FY 11	Interrupt inputs 4th INT byte

Examples of free configuration:

a) 16 inputs/16 outputs

Initial address I:		DW n+4
Initial address Q:		DW n+5

b) 32E

Initial address I:	1st word	DW n+4
Initial address Q:	2nd word	DW n+5

c) IP 65

Initial address I:		DW n+4
Initial address Q:		DW n+5

Caution:

Only the 1st byte is configured in each case; the subsequent bytes are used with consecutive numbering (special treatment).

d) DMP compact

Can be issued per input or output module (depending on complement).

e.g. Submodule 1	either	DW n + 4 (if input)
	or	DW n + 5 (if output)
	contains byte address	

e) Mini/Maxi EU

Identified available (value -1) in DW n+4 and DW n+5.

Configuration notes for various module types

DMP 16 inputs/16 outputs

IP65 (same as 16 inp./16 outp.)

DW n	Module status	UN/UB
DW n+1	Interface No.	1/3
DW n+2	MPC No.	1/2
DW n+3	Terminal block No.	1...15
DW n+4	Initial address inputs	0...255
DW n+5	Initial address outputs	0...255
DW n+6	Unassigned	-1
:		
DW n+19	Unassigned	-1

DMP 32 inputs

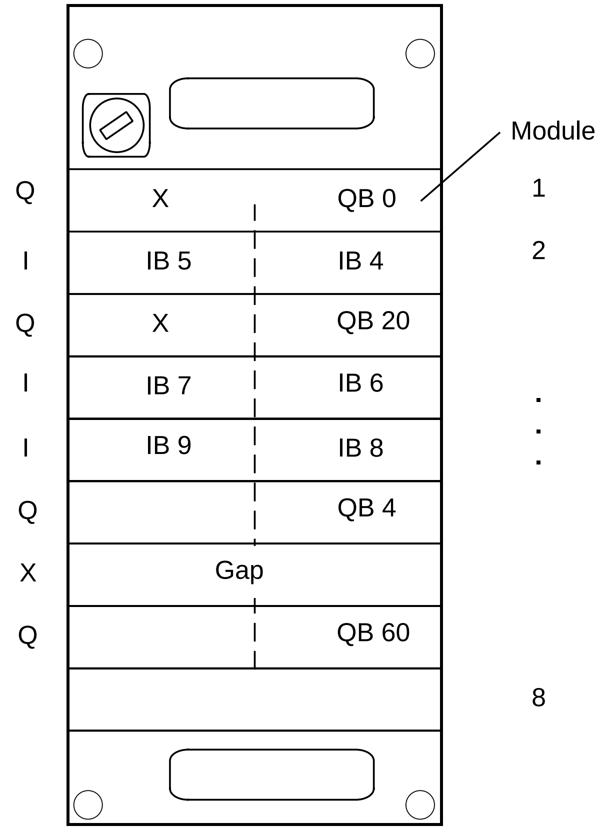
DW n	Module status	UN/UB
DW n+1	Interface No.	1/3
DW n+2	MPC No.	1/2
DW n+3	Terminal block No.	1...15
DW n+4	Initial address inputs 1st IW	0...255
DW n+5	Unassigned	-1
DWn+6	Initial address outputs 2nd IW	0...255
DW n+7	Unassigned	-1
:		
DW n+19	Unassigned	-1

DMP compact
(occupies 24 input/output bytes)

DW n	Module status	UN/UB
DW n+1	Interface No.	1/3
DW n+2	MPC No.	1/2
DW n+3	Terminal block No.	1...15
DW n+4	Initial address inputs 1st module	0...255
DW n+5	Initial address outputs 1st module	0...255
DW n+6	Initial address inputs 2nd module	0...255
DW n+7	Initial address outputs 2nd module	0...255
:		
DW n+18	Initial address inputs 8th module	0...255
DW n+19	Initial address outputs 8th module	0...255

(8 output modules occupy a whole output word; the high-order byte of the output word cannot be used.)

Example: DMP compact station with free configuration



	Station No.	Switch position	Input/output addresses
DMP comp. term.bl.	2	D	
1st module			QB 0
2nd module			IW 4
3rd module			QB 20
4th module			IW 6
5th module			IW 8
6th module			QB 4
7th module			X gab
8th module			QB 60

Unlike machine data configuration, fixed slot and address configuration is not necessary here. The modules are word addressed. Gaps are allowed.

Configuration DB/DX without interrupt processing

0	KC = ”UN”;	UN = enable / UB = disable of terminal block
1	KF = + 00003;	Interface DMP module No. 3
2	KF = + 00001;	MPC 1
3	KF = + 00001;	Terminal block 1; DMP 16 inputs/16 outputs rotary switch "E"
4	KF = + 00070;	Inputs from IB 70
5	KF= + 00084;	Outputs from QB 84
6	KF = - 00001;	
7	KF = - 00001;	
8	KF = - 00001;	
9	KF = - 00001;	
10	KF = - 00001;	
11	KF = - 00001;	
12	KF = - 00001;	
13	KF = - 00001;	
14	KF = - 00001;	
15	KF = - 00001;	
16	KF = - 00001;	
17	KF = - 00001;	
18	KF = - 00001;	
19	KF = - 00001;	
20	KC = ”UN”;	UN = enable / UB = disable
21	KF = + 00003;	Anschaltung Interface-DMP Nr. 3
22	KF = + 00001;	MPC 1
23	KF = + 00002;	Terminal block 2; DMP compact station rotary switch "D"
24 25	KF = - 00001; KF = + 00000;	QB 0
26 27	KF = + 00004; KF = - 00001;	IW 4
28 29	KF = - 00001; KF = + 00020;	QB 20
30 31	KF = + 00006; KF = - 00001;	IW 6
32 33	KF = + 00008; KF = - 00001;	IW 8
34 35	KF = - 00001; KF = + 00004;	QB 4
36 37	KF = - 00001; KF = - 00001;	Gap
38 39	KF = - 00001; KF = + 00060;	QB 60

Preset all data that are not required with "-1".

DX 200 with interrupt processing

•
•
•

40	KC = ”iN”;	Interrupt processing IN = switch on, IB = switch off
		Address IB 0 - 255, -00001 = switch off
41	KF = 6	1st interrupt byte
42	KF = –00001;	2nd interrupt byte
43	KF = –00001;	3rd interrupt byte
44	KF = –00001;	4th interrupt byte
		Edge interpretation 0 edge = 0 1 1 edge = 1 0
45	KM = 11001111 00000000	1st byte **/** 2nd byte
46	KM = 00000000 00000000	3rd byte **/** 4th byte
		Enable bit = 1, disable bit = 0
47	KM = 11111111 00000000	1st byte **/** 2nd byte
48	KM = 00000000 00000000	3rd byte **/** 4th byte

8 Machine Interface

8.1 Actual-value input measuring circuit (6FX1121-4B.. module)

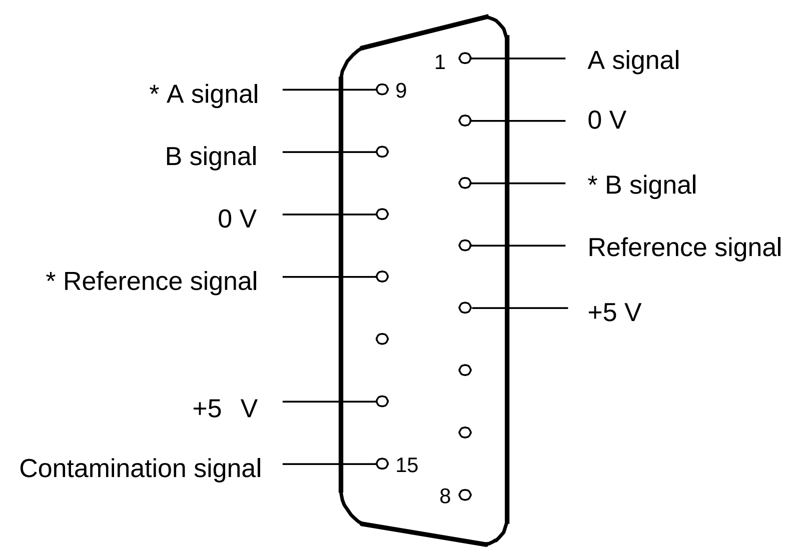
8.1.1 Connector pin assignments

The actual values are supplied to the CNC via 15-way connectors (measuring circuit module 6FX1 121-4B...).

1st	actual-value connector	encoder 1
2nd	actual-value connector	encoder 2
3rd	actual-value connector	encoder 3

Incremental rotary encoders (e.g. ROD 426) for linear axes or incremental linear encoders with EXE external pulse-shaper electronics (e.g. LS703 and EXE603 linear scales) are connected.

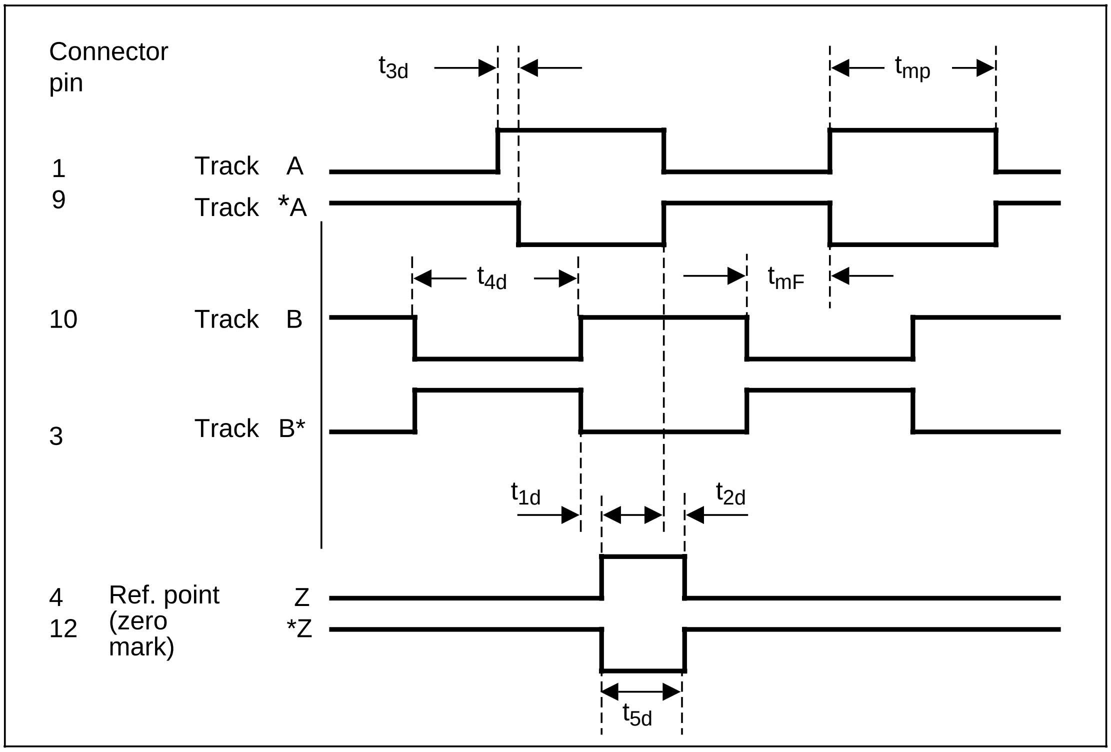
The input signals to the measuring-circuit modules are identical for both encoders. The 6FX1 121-4B. . module is also available with integrated EXE. In this case the signals are routed from the measuring head directly to the measuring-circuit modules and are converted at the module into TTL signals (also see Section 8.1.5).



Actual value connector pin assignments

8.1.2 Differential input

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

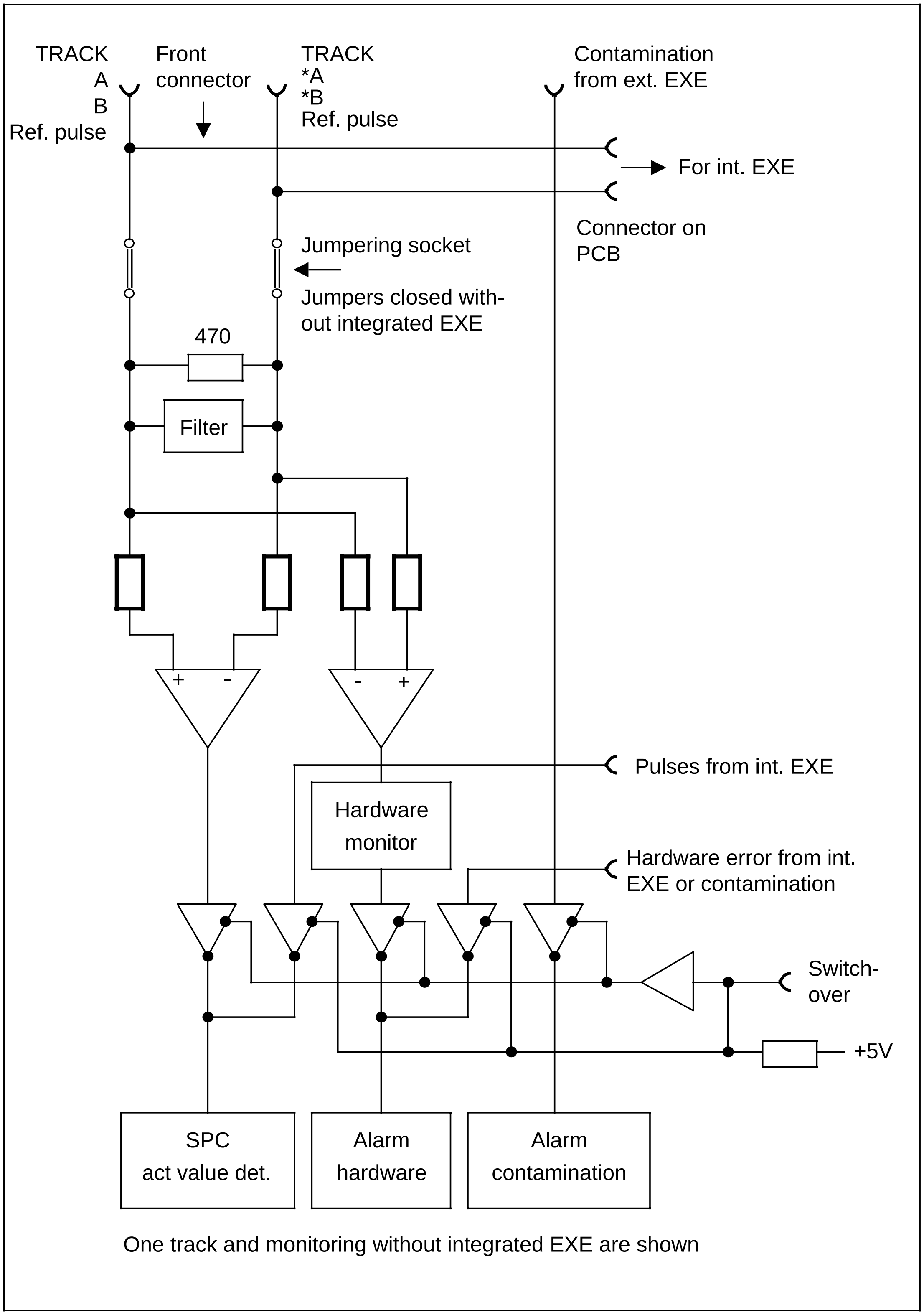


Input signals for digital measuring systems with differential output

Some important characteristics:

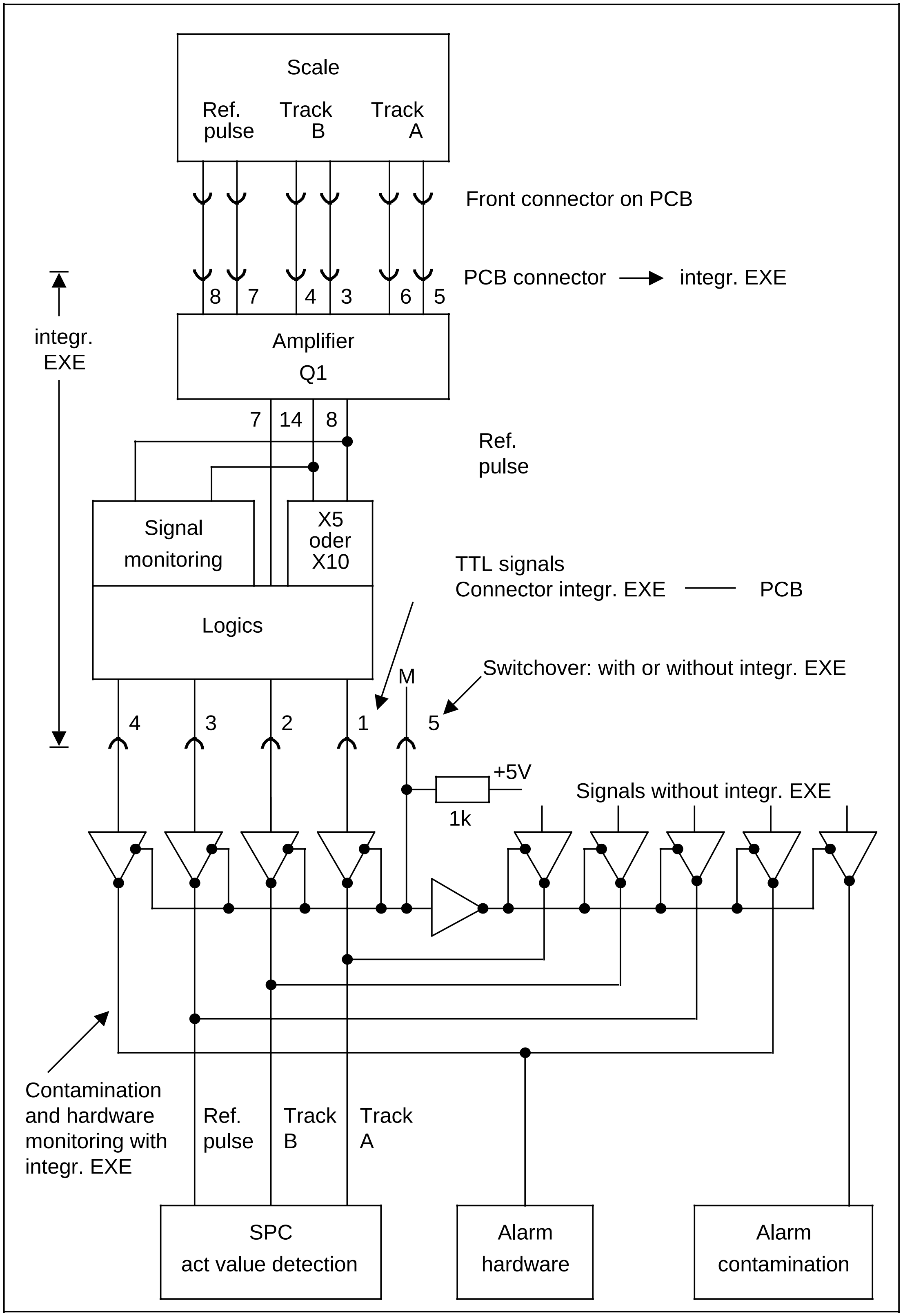
• Encoder supply voltage	5 V ± 5 %
• Ripple of the power supply	100 m V_{pp}
• Current per encoder system	300 mA
• Ohmic input resistance	470 Ohm
• Dynamic input resistance	110 Ohm
• Differential input voltage e.g. between A and *A	1 V
• Differential input voltage max.	10 V
• Maximum input frequency at 90° phase displacement between A and B track pulses	1 MHz (without EXE) 300 kHz (with EXE)
• Minimum pulse width t_{mP}	400 ns
• Minimum edge spacing (change)	350 ns
• Noise immunity (DIN 57847) - noise signal width	3 kV
• Maximum cable lenth to the encoder using SINUMERIK cables	35 metres
• Minimum spacing between two successive edges t_{mF}	200 ns
• t_{1d} and t_{2d}	60 ns
• Maximum spacing between two successive edges of a track t_{3d}	20 ns
• Condition for reference track (zero mark)	Z = high, if A and B= high
• Rate of change (all signals)	1 V / µs
• Minimum length of the zero mark t_{5d}	200 ns

8.1.3 Equivalent circuit diagram with differential input



Equivalent circuit diagram with differential input

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

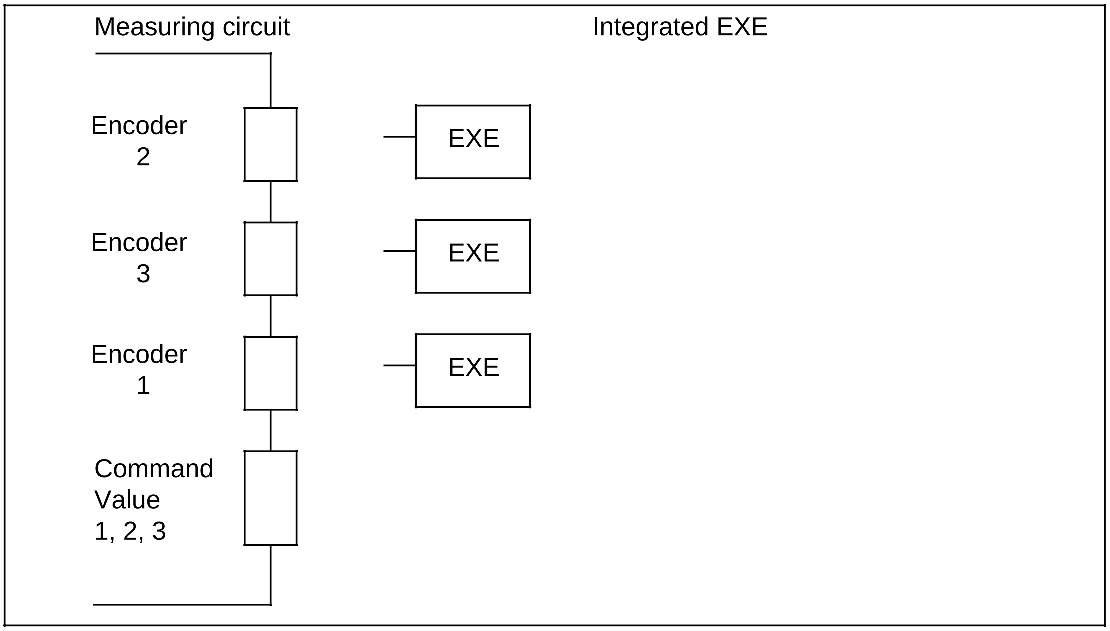


Equivalent circuit diagram for actual-value input with integrated EXE

8.1.5 Complement with integrated EXE

A complement of up to three EXEs is possible with the 6FX1 121-4B... measuring circuit module. Each of these tree EXEs can be jumpered as a 5-fold or 10-fold E (factory jumpering: 5-fold).

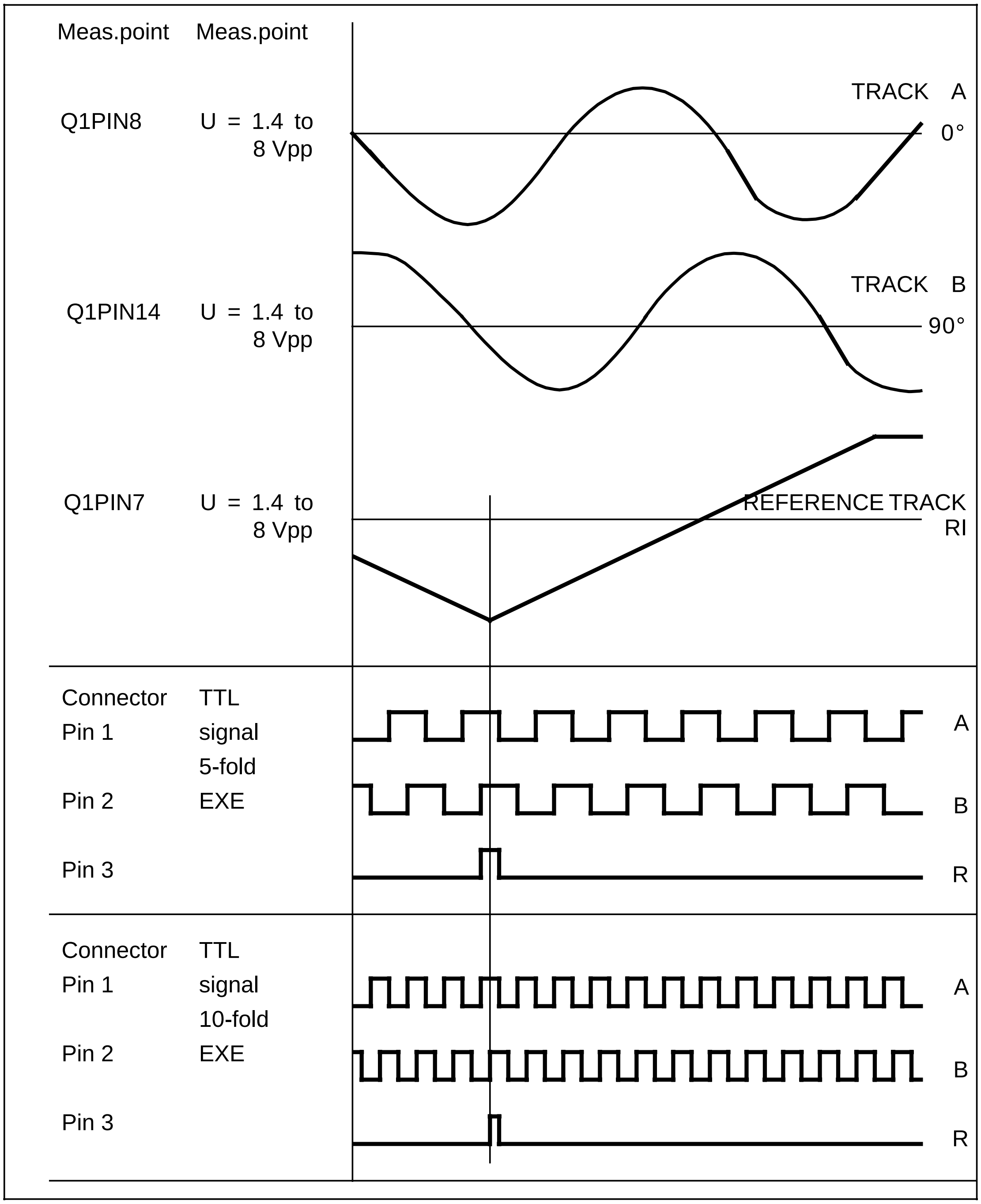
The permissible combinations with the maximum complement are as follows:



Different complements with integrated EXEs

Width	Order No.	Order No.	Axis 1 X 131	Axis 2 X 111	Axis 3 X 121
20 mm	6FX1 121-4BA01	K 20			
40 mm	6FX1 121-4BB01	K 21			
40 mm	6FX1 121-4BL02	K 23	EXE		
40 mm	6FX1 121-4BM02	K 24	EXE	EXE	
40 mm	6FX1 121-4BN02	K 25	EXE	EXE	EXE

8.1.6 Input signals with integrated EXE



Input signals with integrated EXE

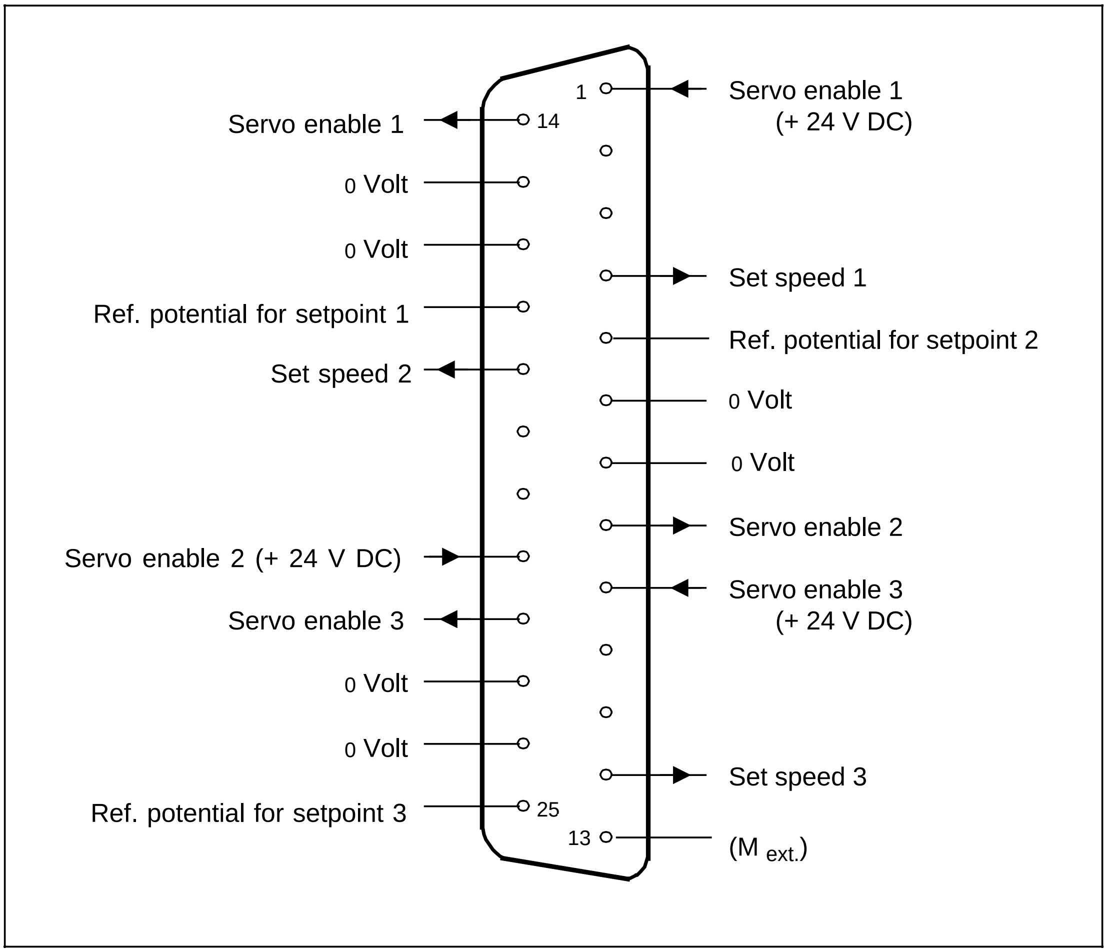
Additional data

- Measurements of Q1 signals performed earth-free with respect to measuring point Uo.
- Phase angle 90° ± 12° of 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
- Reference track approx. 3.5 µA

8.2 Setpoint output measuring circuit (6FX1121-4B.. module)

8.2.1 Connector pin assignments

The set speeds and axis-specific servo enable signals are output from the control via a 25-way connector.

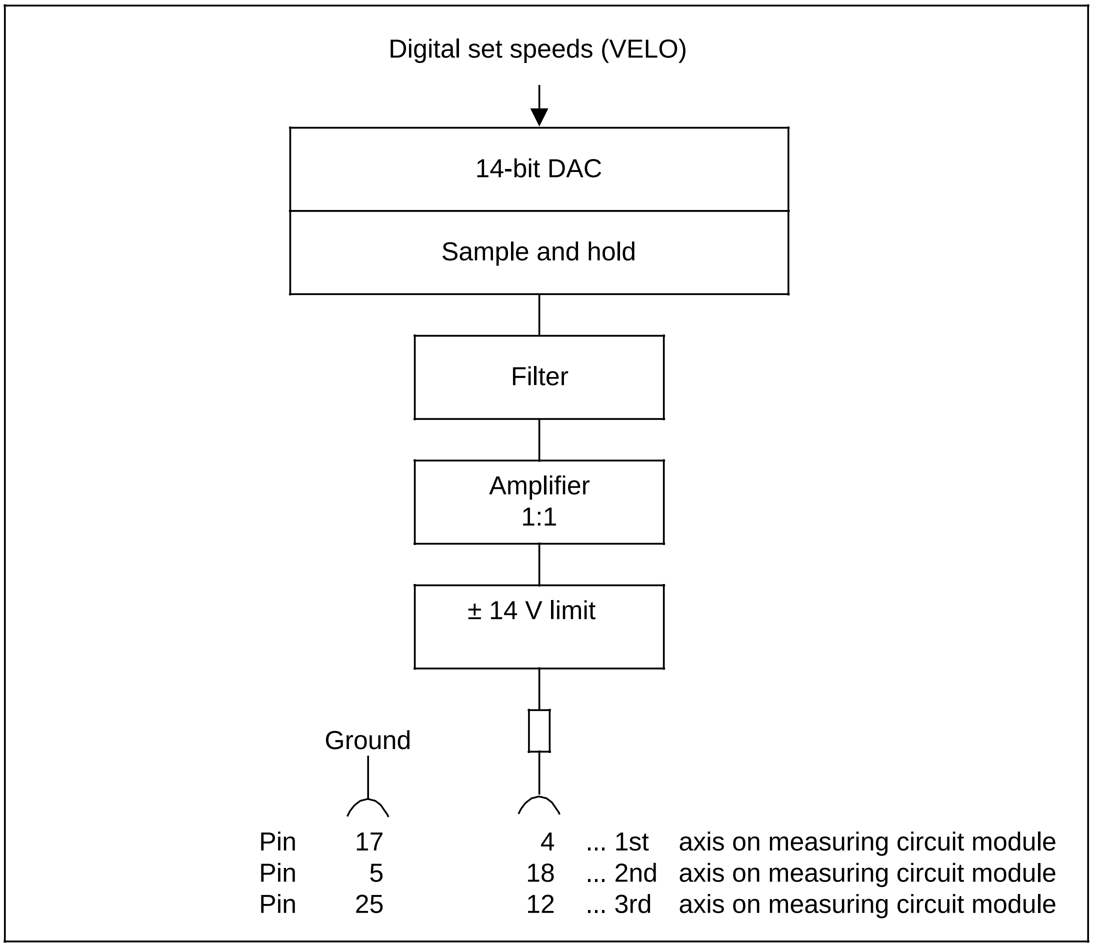


Set value connector pin assignments

Characteristics:

- Analog setpoint
  - Voltage: Max. ± 10 V (hardware limit ±14 V)
  - Current: Max. 2 mA
- Servo enable
  - Supply voltage: 20 to 30 V DC incl. ripple
  - Current: Max. 100 mA (short-circuit proof)
  - Non-floating switching transistor up to version E
  - Floating contact relay from version F.

8.2.2 Equivalent circuit diagram of the set speed output (for one axis)



Equivalent circuit diagram of the set speed output

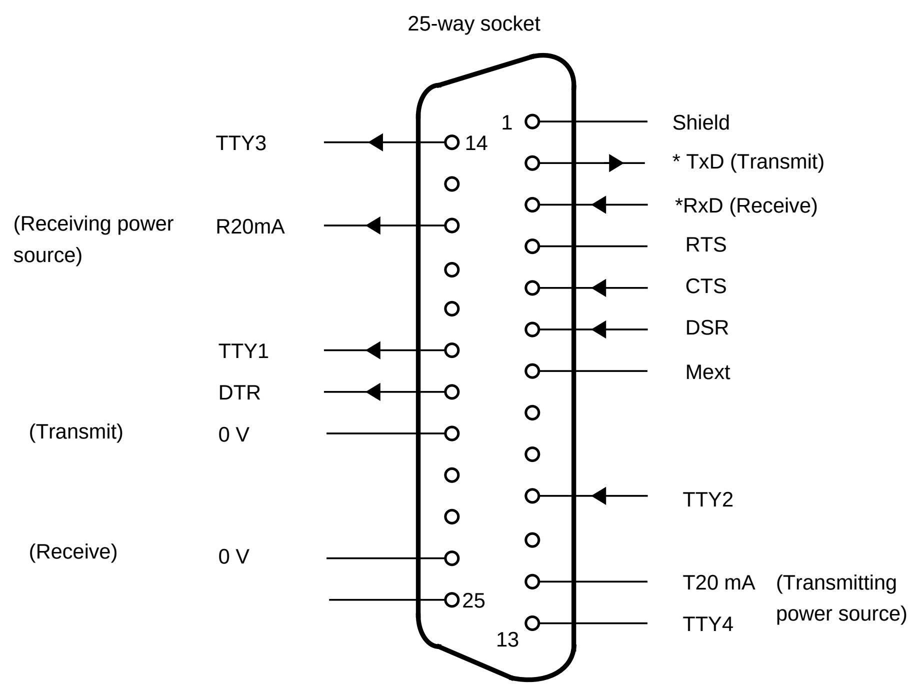
8.3 Serial interface (V.24 (RS232C) + 20 mA)

The SINUMERIK 880 GA2 has up to 4 serial interfaces:

Interface 1 V.24 (RS232C) + 20 mA	OP CPU	X121 (Standard)
Interface 2 V.24 (RS232C)	OP CPU	X131 (Option)
Interface 3 V.24 (RS232C) + 20 mA	COM CPU	X121 (Standard)
Interface 4 V.24 (RS232C)	COM CPU	X131 (Option)

Note:

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 (Configuring Instructions).



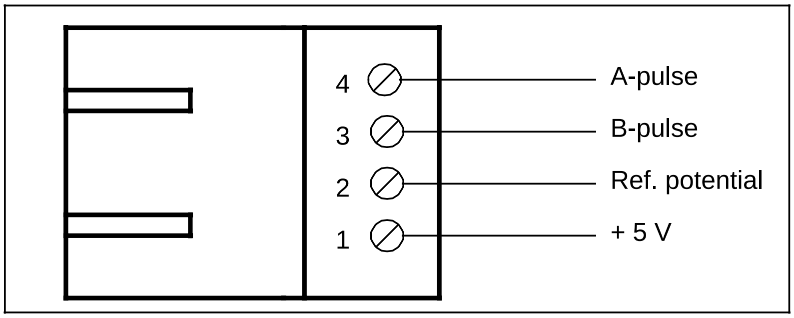
Interface assignment

Characteristics

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

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

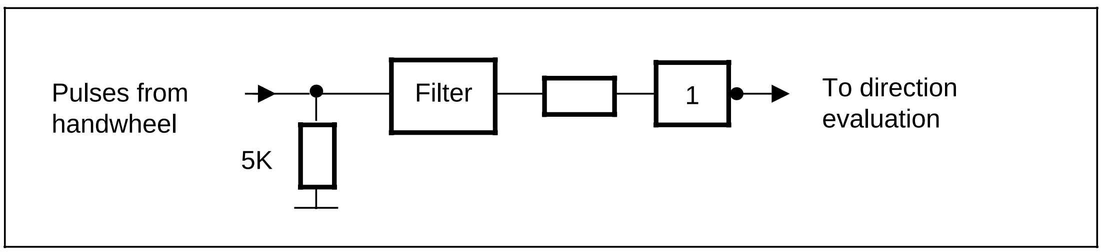
8.4.1 Connector pin assignments



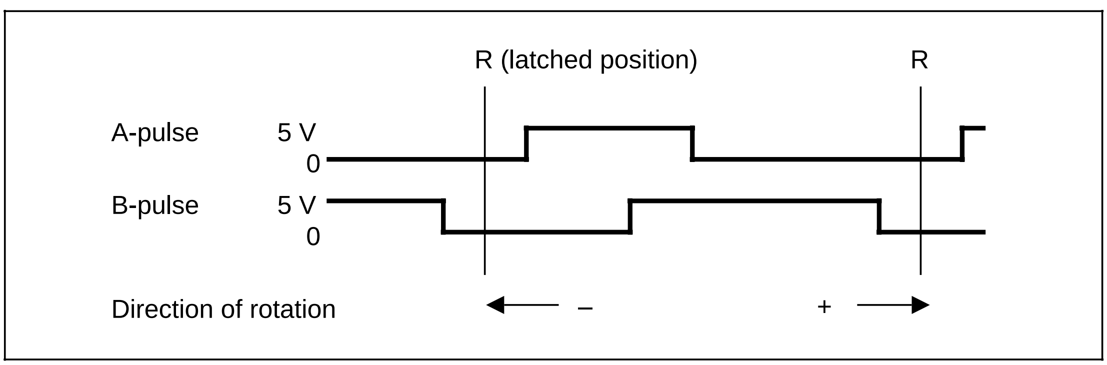
The control can supply up to 150 mA per handwheel at the +5 V pin.

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

8.4.2 Equivalent circuit diagram



Two pulses A and B with 90° offset are output when changing from one latched position to the next (pulse-interval ratio 1 : 1). The outputs are at a low level in the latched positions.

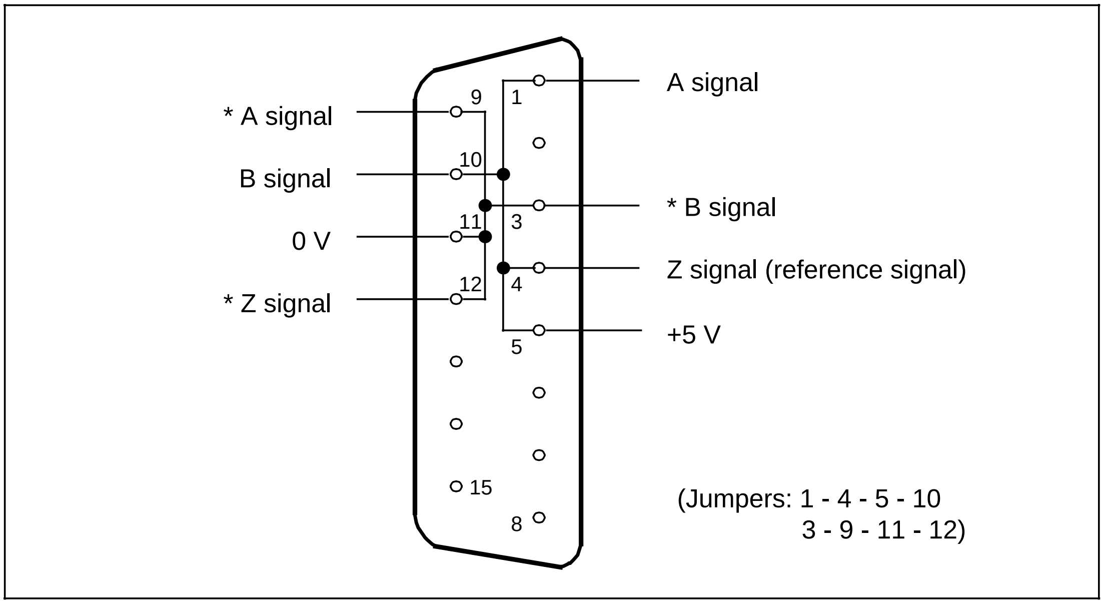


Max. frequency	5 kHz
Low level:	0.9 V
High level:	3.6 V
Voltage:	max. 5.25 V

8.5 Short-circuiting connector for actual-value input

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

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



User-manufacture of connector

9 NC Machine Data/PLC Machine Data/ NC Setting Data

9.1 General

The NC machine data (NC MD) are used to match the NC to the machine tool. They should be carefully determined and optimized during installation, unless the machine manufacturer or end user specifies fixed settings.

The input is made in a memory on the COM CPU which is protected against data loss by the central backup battery when the control is switched off.

The NC MD may be entered in initial clear mode via the V.24 (RS232) interface at any time and in normal operation both manually and via the V.24 interface if the password has been specified.

9.1.1 Control configuration

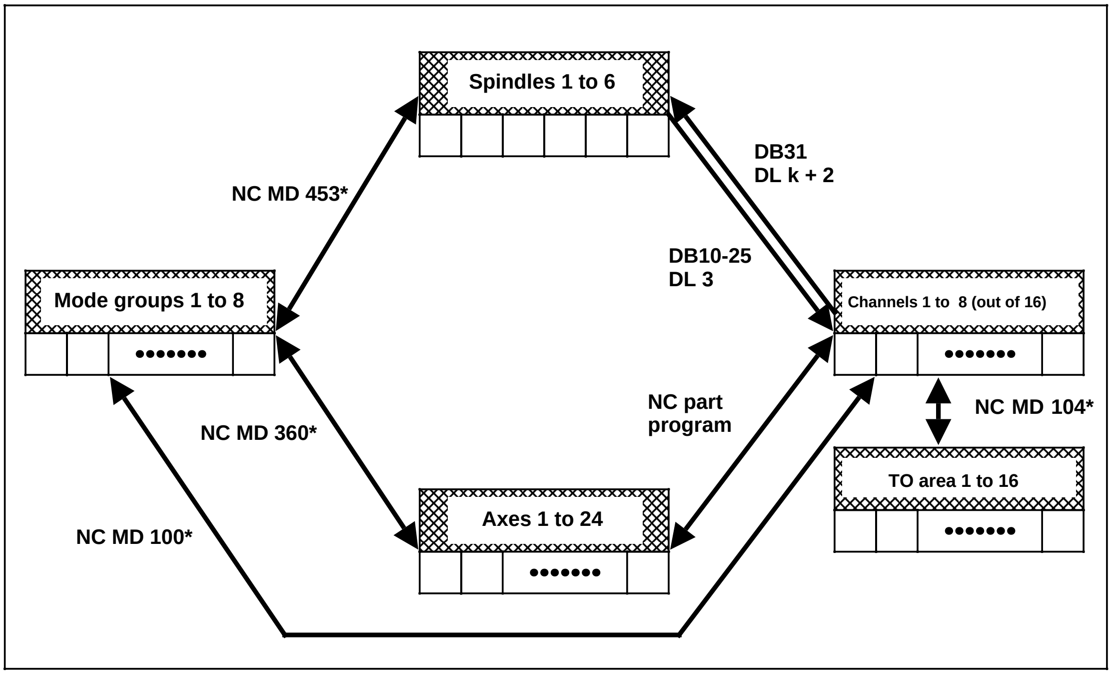
When installing and, to a certain extent, in the course of operation (warm restart) the following parts can be configured flexibly:

- NC CPUs
- Servo CPUs
- Measuring circuit modules
- PLC CPUs
- Input/output modules
- PLC expansion units
- Mode groups
- Channels
- TO areas
- Axes
- Spindles

All of these areas are related to the other areas, some directly and some indirectly.

A distinction is made between the software configuration and hardware configuration of SINUMERIK 880 GA2.

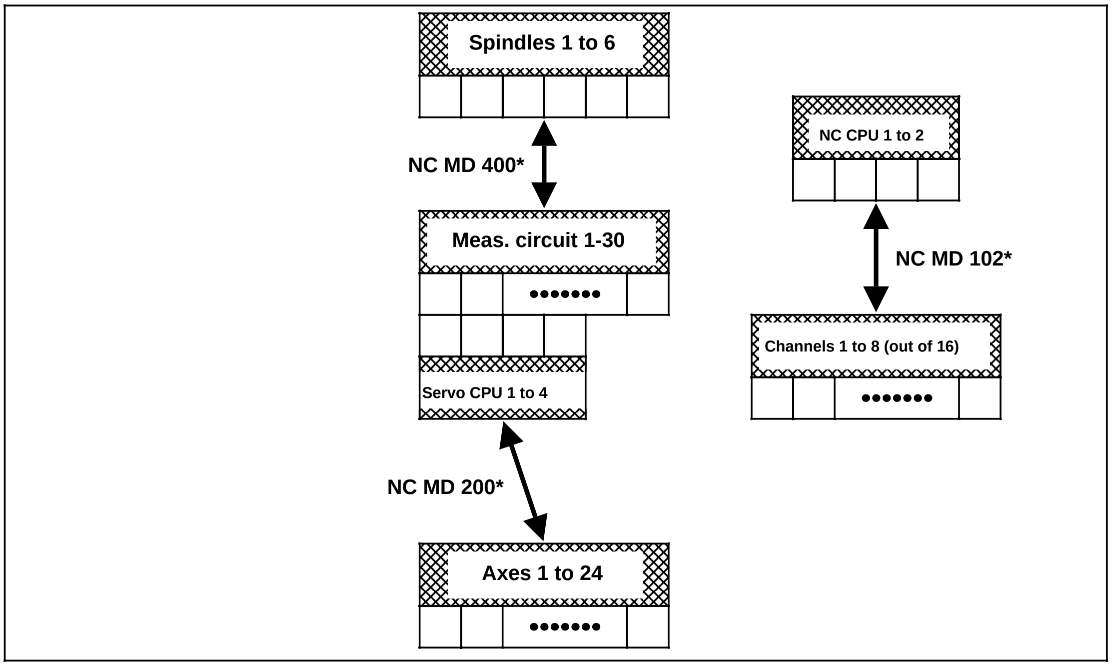
The **software** configuration defines the number of axes and spindles the system possesses and the distribution to the mode groups and channels. The control is adapted to the requirements of the machine.



Note:

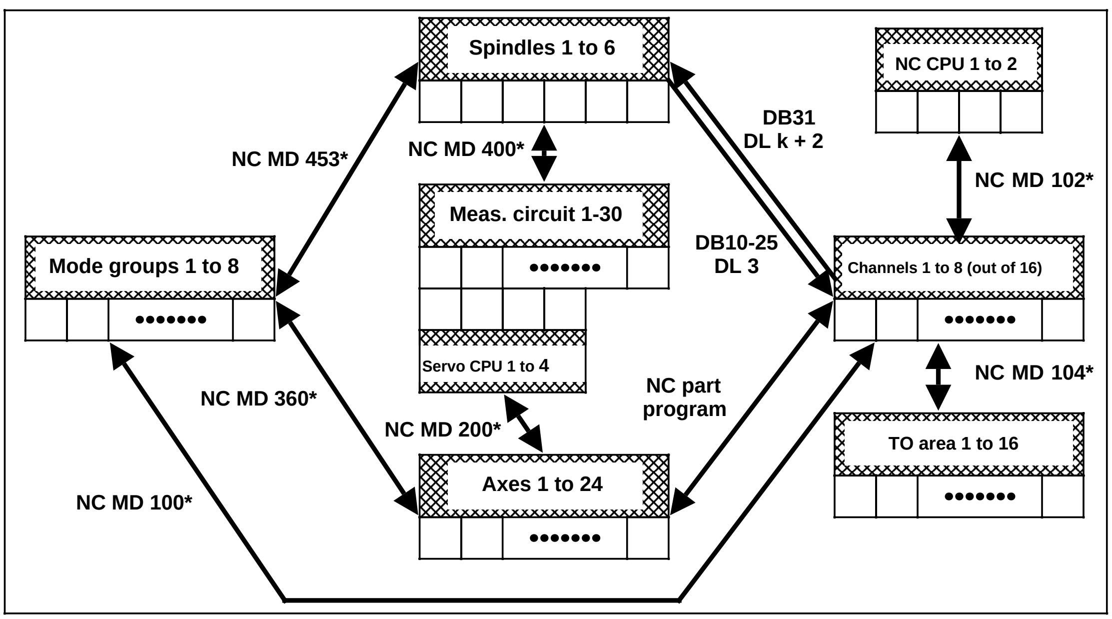
For reasons of compatibility with the SINUMERIK 880 GA1, the maximum possible 8 channels can be selected from the range of 16 channels.

The **hardware** configurations adapt the software to the existing control hardware.
The hardware configuration defines how much CPU power is available (number of CPUs), and how many and which modules must be inserted to attain the required limit data as regards position control scan time and block preparation time.



With the same machines (same software configurations) and same PLC user program it is possible to adapt the required CPU power of the control to the requirements of the customer by hardware configuration.

The hardware and software configurations shown together:



9.1.2 Notes on configuration

Up to eight mode groups and 8 channels are feasible in the SINUMERIK 880 GA2. Machine data is used to specify the assignment of channels to mode groups, i.e. the mode groups are at a higher level than the channels. Machine data is also used to assign the available axes to the mode groups. Up to 30 measuring circuits (24 axes and 6 spindles) are possible. A maximum of 8 channels can be assigned to one mode group.

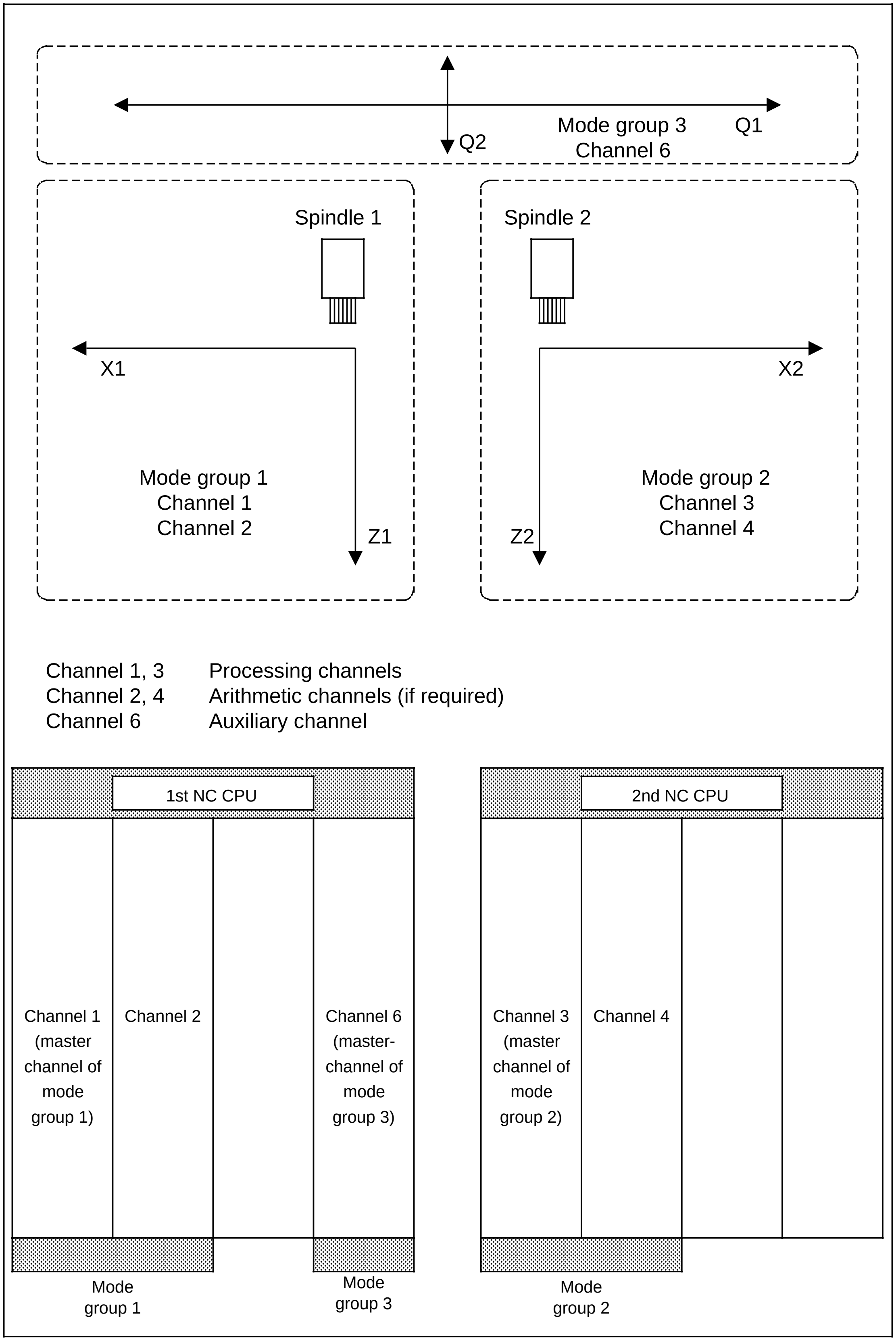
In principle, each channel can be compared with its own NC. Each channel may only process its own program, controlled via the PLC. However, channels belonging to the same mode group may only be operated with the same mode; channels assigned to different mode groups as per the machine data may be operated simultaneously with different modes.

The axes are assigned to the channels in the part program, and not via machine data. This means that an axis can operate in various channels within a given mode group, but not simultaneously.

It is possible to inhibit a given axis in a specific channel (NC machine data) to prevent a collision due to a programming error. The operator can select the various declared mode groups and channels via the operator panel keyboard. At least one channel and one axis must be assigned to each mode group. Mode groups without spindles are permitted.

A channel may only operate the axes in its mode group. An axis may only operate in its assigned mode group. The warm restart function makes it possible to modify the system configuration (assignment to mode groups) without having to re-approach the reference points (see Section 11.6 for warm restart).

On the SINUMERIK 880 GA2, a maximum of 4 channels can be activated per NC CPU. With the maximum number of 2 NC CPUs (depending on the version ordered: 1 to 7), therefore, up to 8 channels can be activated. NC machine data is used to determine which NC CPU a channel operates on or whether a channel exists at all. The more channels are activated on an NC CPU the greater the block preparation time. If especially computationally intensive functions are selected (TRANSMIT, 2D/3D coordinate transformation, 3D/5D helical interpolation, etc.) four channels per NC CPU can no longer be activated (alarm ”NC CPU failed”). So only up to two channels per NC CPU can be implemented with TRANSFORMATIONS. It might be necessary to increase the standard interpolation time in this case as well (NC MD 155).



Typical system configuration (double slide machine with two spindles and loading/unloading device)

Function of master channel

In a mode group the channels are processed in ascending order. The first channel in a group (the one with the lowest number) is declared the master channel of the group in the control. On key reset (RESET on machine control panel), it has the task of executing the following functions:

- Clear all reset alarms specific to mode group
- Clear all axial reset alarms in mode groups
- Synchronize all axes in mode groups
- Clear all spindle-specific reset data in mode groups
- Servo reset for axes in mode groups

The master channel also has a special function at the channel-specific NC-PLC interface (DB 10 - 25):

In the event of the following signals being specified in the master channel of a mode group, they are also valid in all the other channels of the same group:

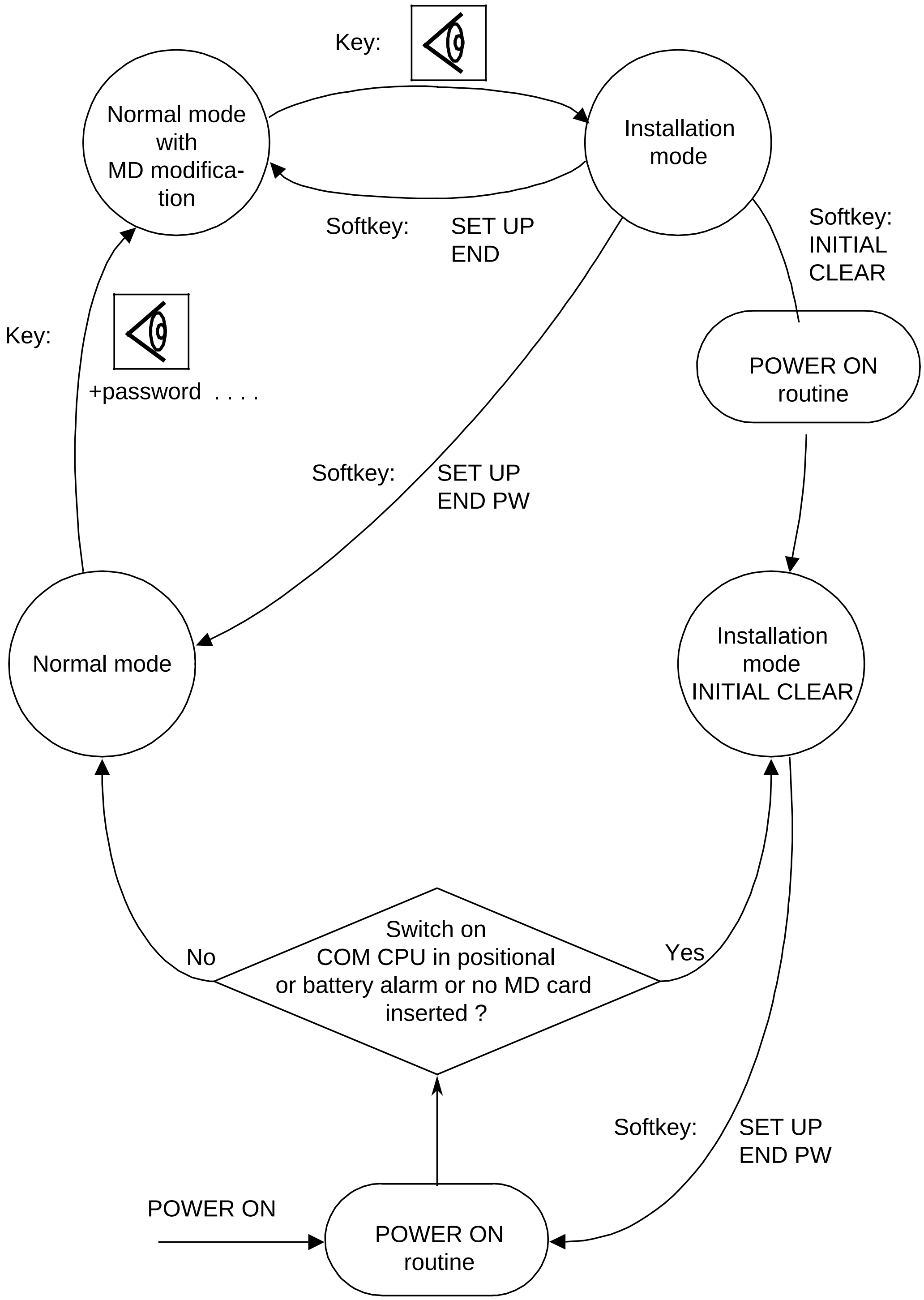
- RESET (DL 0 bit 6)
- Operating mode (DL 0 bits 0 to 3)
- Feedrate override active (DL 1 bit 5)
- Rapid traverse override active (DR 1 bit 5)
- DRF (DL0 bit 7)

If these signals are specified not in the master channel of a mode group, but in another channel of the same group, they are valid only in the channel concerned.

Exception:

RESET and operating mode can be specified only in the master channel, but is valid for the whole mode group.

9.2 Mode selection



9.2.1 Installation mode "INITIAL CLEAR"

Standard installation sequence

In the following installation sequence, it is assumed that neither the NC nor the 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 submodules (6FX1 124-6AA..) must be inserted or connected and correctly jumpered.

The NC MD and NC MD bits important for the PLC are dealt with after the installation sequence.

The installation sequence is mandatory since the NC machine data must be input prior to formatting the user memory and clearing the part program memory.

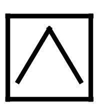
Mandatory sequence:

1. NC initial clear
   Move switch on COM CPU to position 1 (left limit and then 1 position to the right) and operate "POWER ON Reset";
   (press Reset key on power supply unit or switch the system on).
   The control is now in the "INITIAL CLEAR" installation mode.
2. Depress "MACHINE DATA" softkey.

The option bits are already set in the factory so that

points 3 and 4 must be ignored if the control

is in the same state as on delivery.

3. Press "CLEAR NC MD" softkey; NC machine data are cleared.
4. Press "LOAD NC MD" softkey; NC standard machine data are loaded.
5. Press "CLEAR PLC MD" softkey; PLC machine data are cleared.
6. Press "LOAD PLC MD" softkey; PLC standard machine data are loaded.
7. Press "LOAD CYC. MD" softkey; Cycle machine data are loaded.
8. Press "Recall"  key; return to basic display
9. Depress "NC DATA" softkey.
10. Depress "FORMAT USER M. " softkey
    The following data are erased:
    - Tool offsets
    - Setting data
    - R parameters
    - MIB parameters
    - Zero offsets

The following data are preset:

- The setting data for serial interface 3 are preset
- The MIB parameters are formatted according to NC MD 4 and 5
- The tool offset memory is formatted according to the Option bits and NC MD 210 to 226
- The working range limits are preset to max. values
- Spindle speed limit (S100) is preset

11. Press "CLEAR PART PR" softkey; the part program memory is formatted according to Option bits and cleared.

12. Read in PLC and NC machine data if available via the serial interface. In installation mode, the setting data bits for the serial interfaces are selected by depressing the "DATA IN-OUT" softkey and can be matched manually to the external input/output device.

Caution! The option bits must not be overwritten (check!)

Do not read in SINUMERIK 880 GA1 machine data files because the formats of some of the machine data have changed.

13. Press "NC DATA" softkey.

14. Press "FORMAT USER M." softkey.

15. Press "CLEAR PART PR" softkey.

16. Press ”RECALL” key; return to basic display.

17. Press"PLC UTILITY" softkey.

18. Press "PLC INITIAL" softkey.

19. Press"PLC 1 INITIAL" softkey.

20. Press "PLC 2 INITIAL" softkey if 2nd PLC CPU installed.

21. Press ”RECALL” key twice; return to basic display.

22. Move switch on COM CPU to position 0 (left limit).

23. Press "SET UP END PW" softkey POWER ON is performed (see Section 9.2).

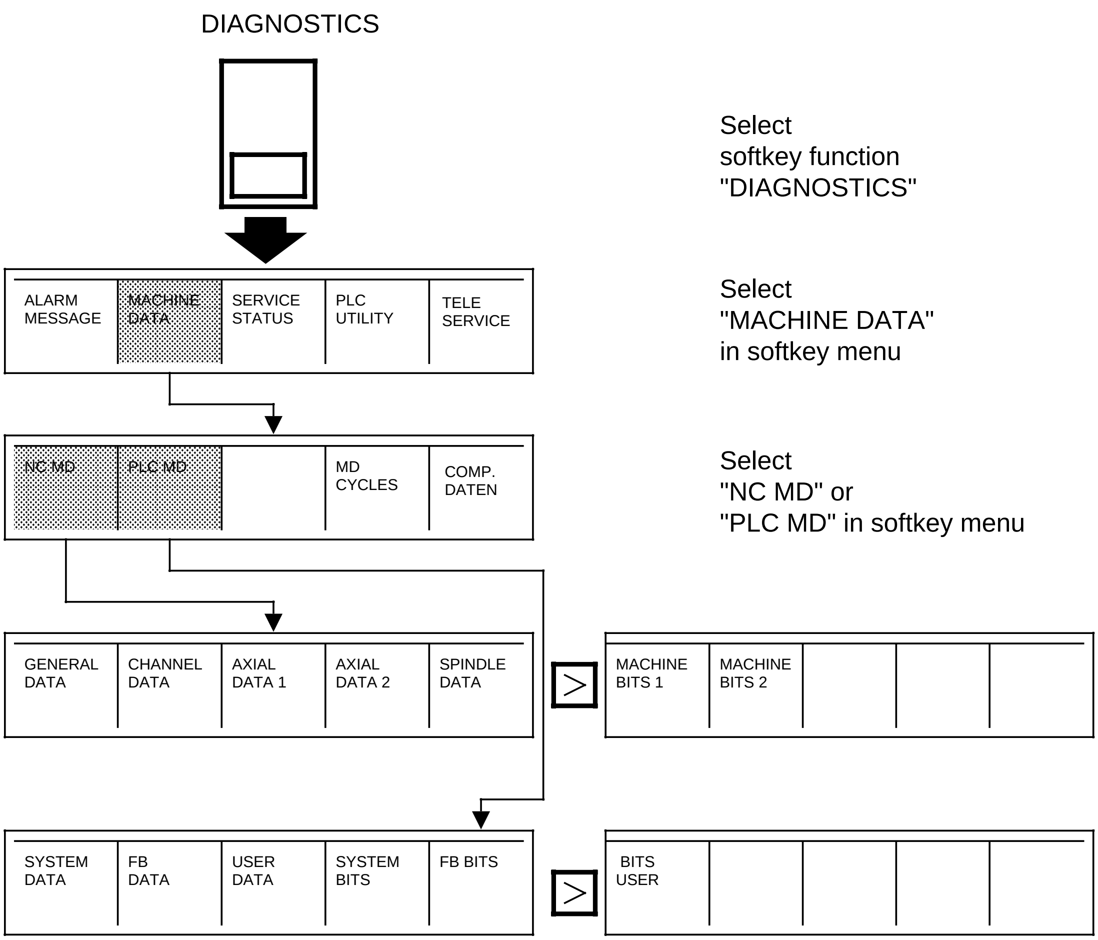
No NC MD and PLC MD may be entered, displayed or edited manually in the installation mode.

During installation the PLC is not in cyclic operation and flag F24.3 is set, so the basic program performs no processing between the interface and the NC.

In view of the fact that no NC MD and PLC MD can be transmitted from the PLC to the NC (FB 62) in the installation mode, the PLC may assume a deliberate stop status. This applies to package 1 (tool management) etc.

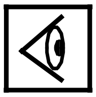
9.2.2 Normal mode (with MD modification)

In normal operation machine data can be read, manually edited and read in via the V.24 (RS232) interface. The machine data are selected using the "DIAGNOSTICS" softkey.

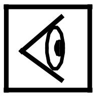


Machine data cannot be edited until the password has been entered.

Enabling MD input

- Press "Start "  key and enter password ... .

Inhibiting MD input

- Press "Start "  key and "SET UP END PW" softkey.

9.2.3 Installation mode

Three functions can be activated in the installation mode

"INITIAL CLEAR" softkey:	Software-triggered hardware reset (POWER ON) executed as shown in Section 9.2, mode selection.
"SET UP END" softkey:	Jump to normal mode with MD modification without cancelling the password.
"SET UP END PW":	Jump to normal mode and cancellation of the password. The machine data cannot be edited without entering the password.

9.3 NC machine data

9.3.1 General

Breakdown of NC machine data:

- NC MD 0 - 999: General values
- NC MD 1000 - 1999: Channel-specific values
- NC MD 2000 - 3999: Axis-specific values
- NC MD 4000 - 4999: Spindle-specific values
- NC MD 5000 - 5199: General bits
- NC MD 5200 - 539*: Spindle-specific bits
- NC MD 5400 - 559*: Channel-specific bits
- NC MD 5600 - 596*: Axis-specific bits
- NC MD 6000 - 6999: Leadscrew error compensation flags
- NC MD 1000* - 1799*: Axis-specific values
- NC MD 1800* - 1999*: Axis-specific bits

Effectiveness of individual machine data:

• configuration-specific data (e.g.: MD 200*, 400*, 156, ...)	after POWER ON
• axis-specific data (e.g.: MD 204*, 240*, ...)	after RESET (mode group-specific)
• spindle-specific data (e.g.: MD 520*, Bit 1, ...)	after NC STOP/NC START
• display-specific data (e.g.: MD 5007, Bit 7, ...)	immediately

A guarantee that **all** machine data will become active immediately according to the function specified here (RESET, NC STOP/START) can **not** be given.

In case of doubt, users are advised to execute the POWER ON routine since this activates **all** NC MDs (exceptions: MD 4, 5, 13, 210 to 226).

This is the case when
a) the NC is switched off and on
or
b) the "SET UP END PW" or "INITIAL CLEAR" softkey is pressed after the MD input in the installation mode.

After editing certain MDs, the memory has to be reformatted.

• NC MD 4	”FORMAT USER M.” softkey
• NC MD 5	”FORMAT USER M.” softkey
• NC MD 13	”FORMAT USER M.” softkey
• NC MD 210 to 226	”FORMAT USER M.” softkey

Caution!

For safety reasons "POWER ON" should generally be performed (switch control off/on) after editing NC machine data.

9.3.2 Position control, input and measuring system resolution

In the case of the SINUMERIK 880 GA2, position control resolution and input resolution may be entered independently. However, in order to ensure a closed position control loop, the pulses from the digital measuring system and the precision of the control must be matched to one another.

The new units of measurement used are "unit (MS)" for position control resolution and "unit (IS)" for input resolution.

The following applies:

1 unit (MS) = 2 units of position control resolution
1 unit (IS) = 1 unit of input resolution

Example:

Given a position control resolution of 0.0005 mm and an input resolution of 0.001 mm, then
1 unit (IS) = 1 unit (MS) = 1 μm

For the input, position control and display resolution see NC MD 5002 and NC MD 1800*.

9.3.3 Input units

Unit (MS) = 2 units of position control resolution (MS reference system)

1 unit of position control resolution = 1/2 μm
(MD 1800* = _ _ _ _ _ 010) 1 unit (MS) = 1 μm

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

1 unit of input resolution = 1 μm
(MD 5002 = _010_ _ _ _) 1 unit (IS) = 1 μm

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

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

9.3.4 Other machine data

All machine data and machine data bits not described in the Installation Guide must be set to **ZERO**.

9.3.5 NC MD description

To maintain clarity in spite of the large number of machine data, the MDs for axes, spindles and channels are marked with a "*". This "*" indicates exactly **which** axis, spindle or channel is being refferred to, whereby the "*" is replaced by a consecutive number starting with 0.

"*"	0	Axis 1,	Spindle 1,	Channel 1
	1	.	.	.
	2	.	.	.
	3	.	.	.
	4	.	.	.
	5	.	Spindle 6	.
	.			
	.	.		.
	.	.		.
	15	.	Channel 16 (max. 8 can be activated)	
	.	.		
	.	.		
	.	.		
	23	Axis 24		

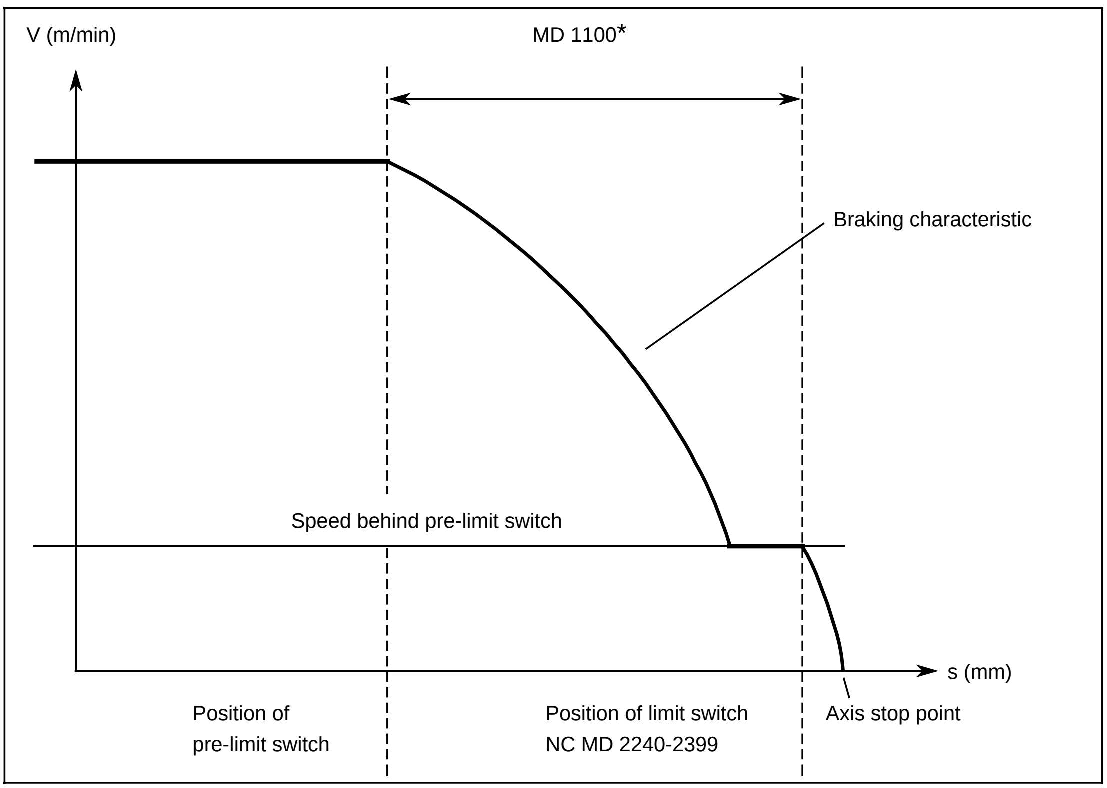
Note:

The user can activate only 8 of the available 16 channels.
Example: channels 1,2,3,5,9,10,15,16.

9.3.5.1 General values

1	Speed behind prelimit switch		1
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**100 000**	**1000 units / min (IS)**

Active: Immediately



Note:

NC MD 1 has no effect if the value 0 is entered in NC MD 1100*.

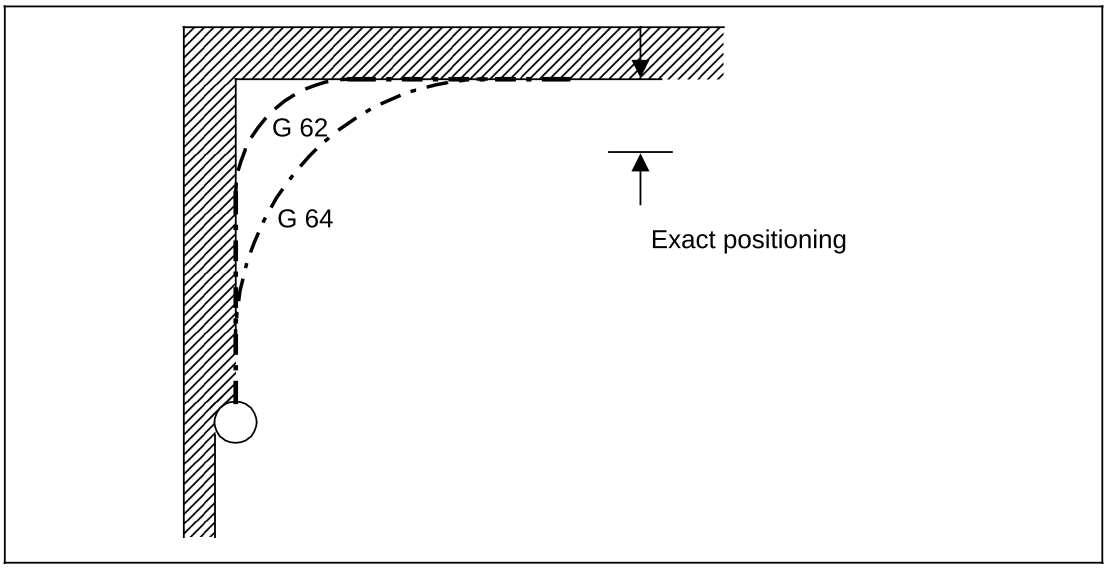
3	**Corner deceleration rate**		**3**
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**100 000**	**1000 units / min (IS)**

Active: Immediately

In continuous-path mode (G64), block transitions are covered without any feedrate reduction, i.e. the tool path is rounded at points of discontinuity over the contour as a function of the traversing rate.

As a result of function G62 (contouring with feedrate reduction), the tool path feedrate is reduced to the rate entered in NC MD 3, provided the selected feed was higher.

The radiusing is thus reduced at discontinuous block transitions.



4	MIB parameters for standard cycles (Siemens)		4
Standard value	Lower input limit	Upper input limit	Units
450	**450**	**500**	**-**

Active: After user memory has been formatted

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

The MIB parameters are required for input of variables for the cycles and blueprint programming if entered on the basis of the drilling and milling patterns or contour definition patterns at the control. NC MD 4 refers to the standard cycles or patterns. If standard cycles and patterns are used, NC MD 4 must be defined with the value 450. The following applies:

NC MD 4	=	500	0	MIB parameters
NC MD 4	=	450	49	MIB parameters

The value to be entered is specified by Siemens AG.
The entered value must not coincide with the value in NC MD 5.

Caution!

Any modification to this value is not valid until the user memory has been formatted ("FORMAT USER M." softkey).

5	MIB parameters for customer		5
Standard value	Lower input limit	Upper input limit	Units
200	**100**	**500**	**-**

Active: After user memory has been formatted

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

The MIB parameters are required for input of variables for the cycles and blueprint programming which the customer has generated himself using the NC workstation, e. g.

- Reference plane
- Drilling/boring depth
- Final machining allowance
- Chamfer etc.

The machine manufacturer must specify how many MIB parameters he requires. The following applies:

NC MD 5	=	100	0	MIB parameters
NC MD 5	=	200	100	MIB parameters
NC MD 5	=	449	349	MIB Parameters

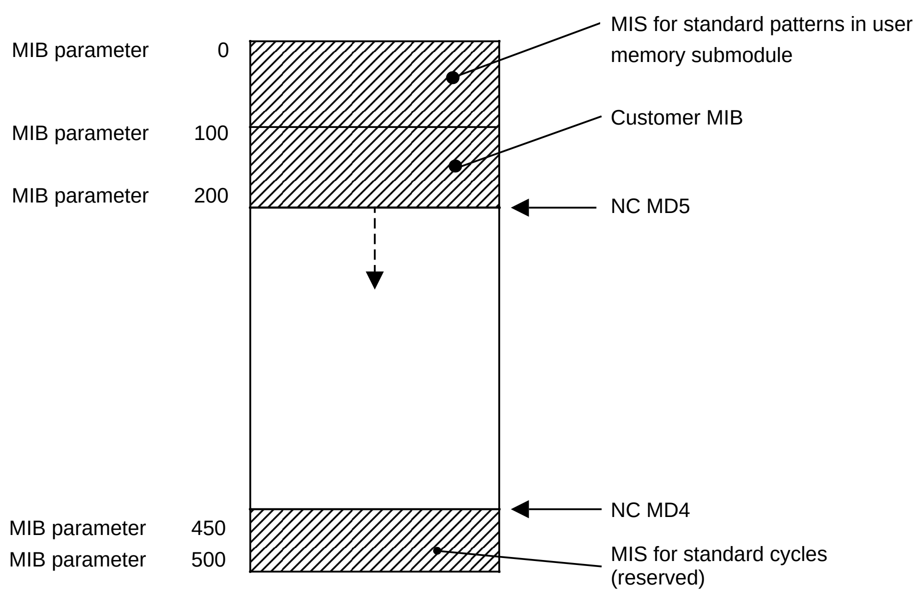
The entered value must not coincide with NC MD 4.

Caution!

Any modification to this value is not valid until the user memory has been formatted ("FORMAT USER M." softkey).

Example:

NC MD4 ... 450
NC MD5 ... 200

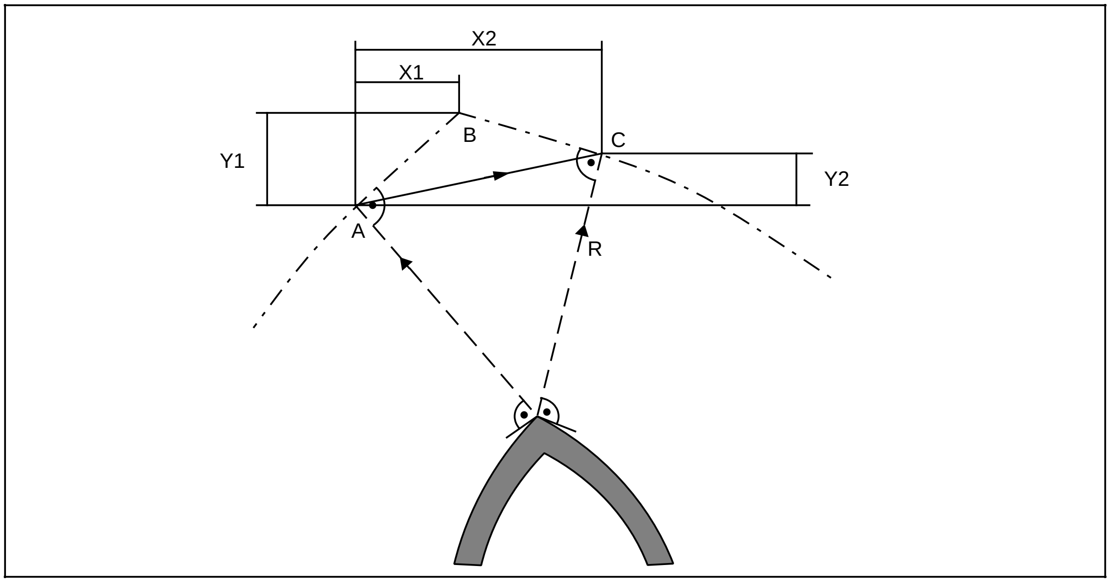


6	Threshold for CRC insertion blocks		6
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**2 000**	**units (IS)**

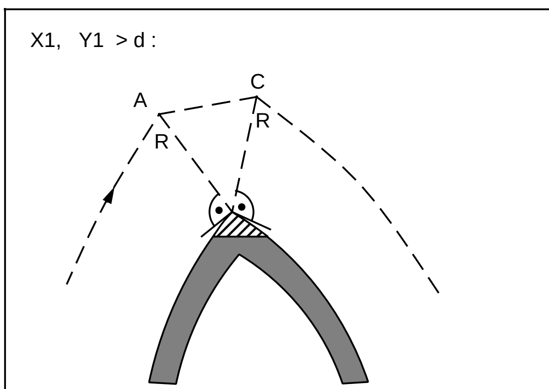
Active: In next block

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 Guide). With these compensating movements, the programmed feedrate is maintained along the cutter centre path; whereas during machining, the feedrate 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 threshold "d" are shortened or omitted as follows:

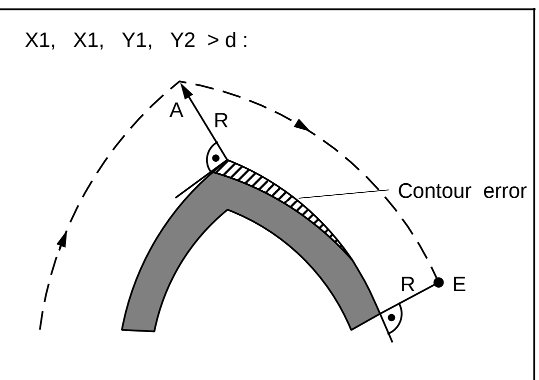
Full self-adjustment:



Shortened self-adjustment A C:



No self-adjustment:

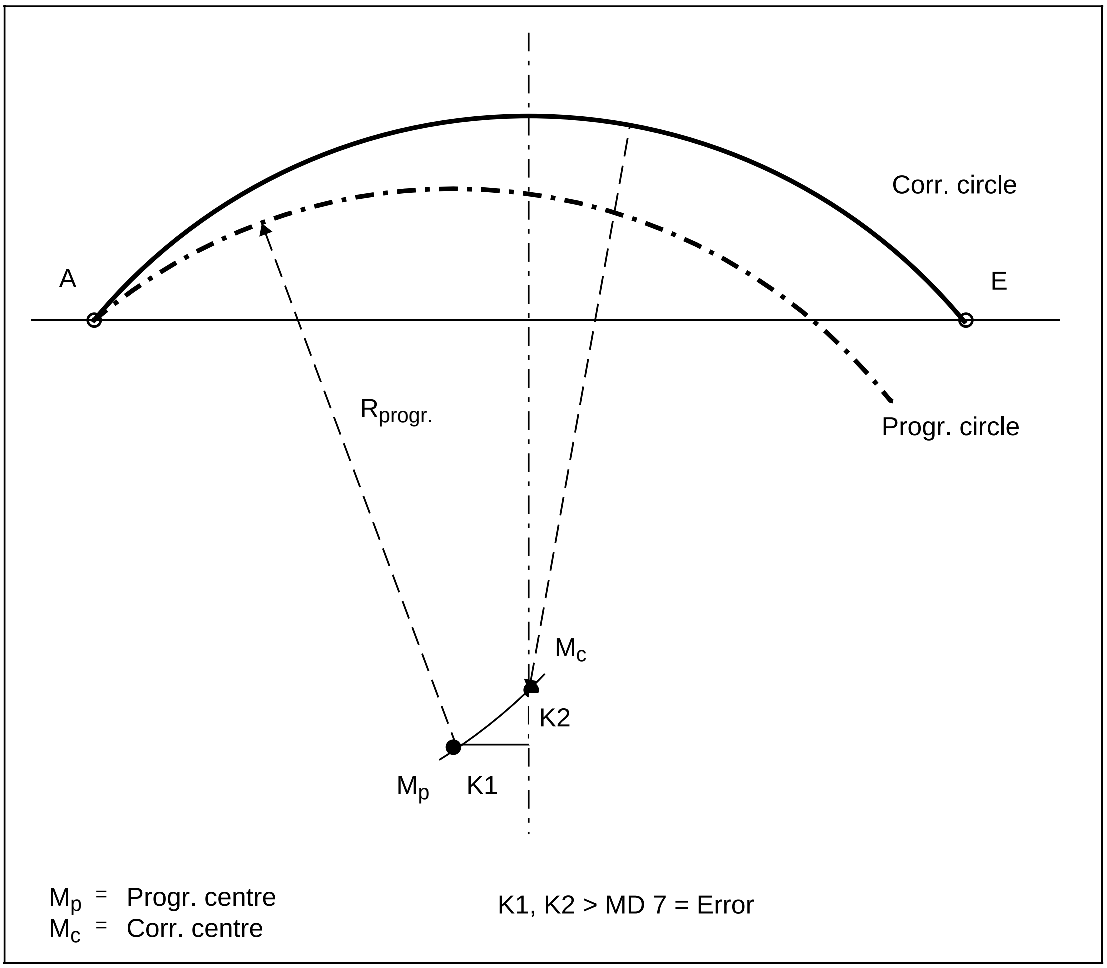


7	Circle end position monitoring		7
Standard value	Lower input limit	Upper input limit	Units
5	**0**	**32 000**	**units (IS)**

Active: In next block

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 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.



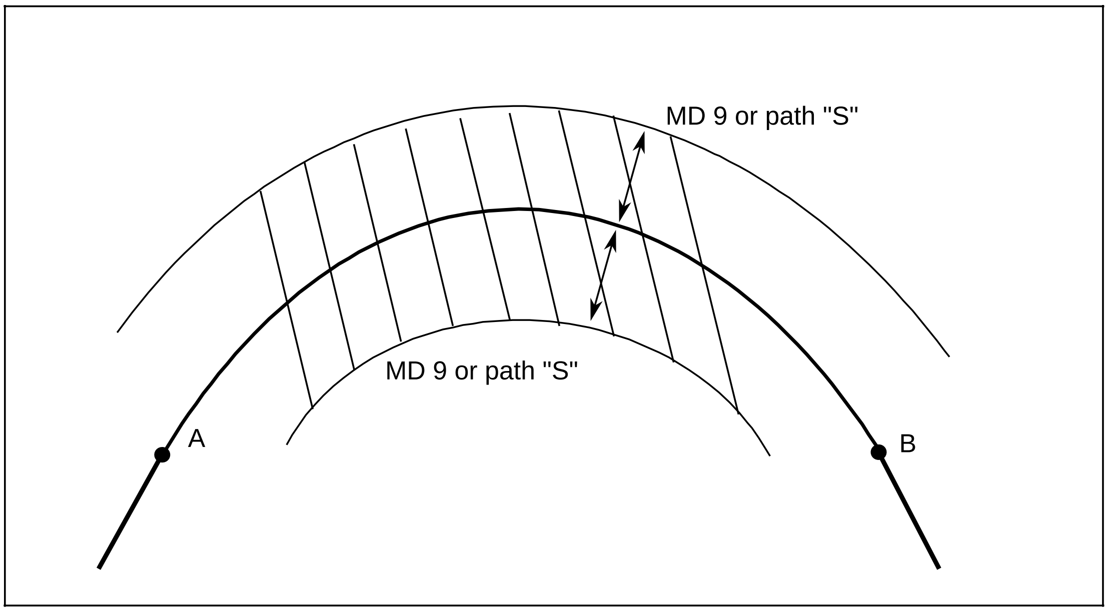
Circle end position monitoring

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

Active: After NC START

Automatic interrupt during circular machining (G2/G3) is followed by departure from the contour in JOG mode.
Repositioning is required in JOG mode prior to NC restart.

Alarm 3018 is output and the program is not started if the axis after NC start is outside the tolerance entered in NC MD 9 (hatched area).



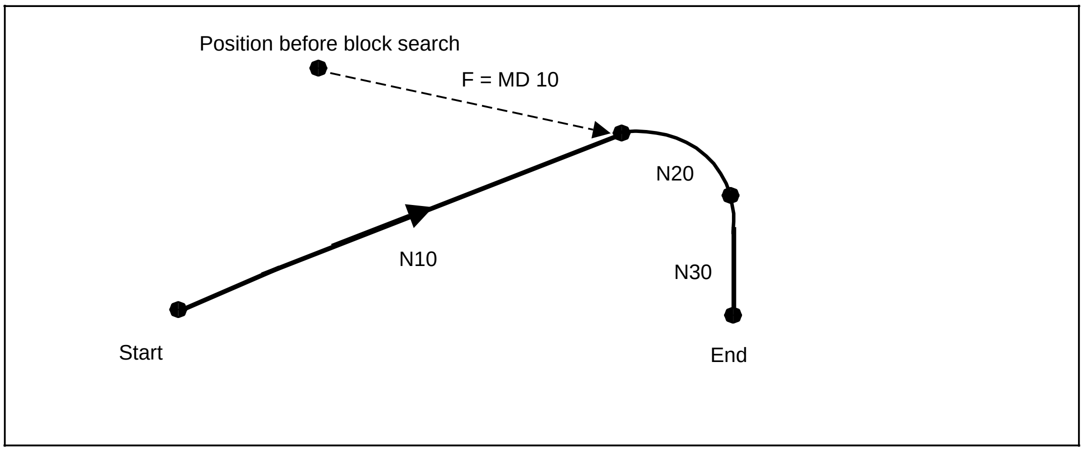
Tolerance range defined in NC MD 9

10	Feed after block search		10
Standard value	**Lower input limit**	**Upper input limit**	**Units**
1 000	**0**	**100 000**	**1000 units / min (IS)**

Active: In next block

If the machining program is not initiated at the start but rather at an existing block using block search, the programmed rate may not be suitable for the traversing path (G95 or G96 active, but no spindle rotation; very small F value programmed).

If the axes are not at the starting point of the first block selected by block search but instead are in some other position, the control would output the programmed speed of the travel block. This could possibly result in critical situations; consequently, the value entered under NC MD 10 is used for travel after block search as far as the first starting position (only with G94). The speed input applies to a feedrate override of 100 %.



Diagrammatic example

Caution!

MD 10 is not active in the event of block search to a block with G00.

The programmed feedrate from the part program is always used to traverse to the end point of the target block if value 0 is entered in NC MD 10.

13	Number of tool offset parameters		13
Standard value	Lower input limit	Upper input limit	Units
10	**10**	**32**	**-**

Active: After user memory has been formatted

Each tool offset has as a standard feature up to ten tool offset parameters (TO parameters) with fixed allocation. These parameters may be expanded by the user to 32 as and when required. These expanded TO parameters (10 to 32) must be allocated by the user using the NC workstation. As the number of TO parameters assigned to a tool offset increases, so the number of tool offsets available to the user decreases since a fixed memory area is assigned to the tool offsets as a whole.

Parameter	Tool offset		Remarks
10	204	409	Reset
11	186	372	
12	170	341	
13	157	315	
14	146	292	
15	136	273	
16	128	256	
17	120	240	
18	113	227	
19	107	215	
20	102	204	
21	97	195	
22	93	186	
23	89	178	
24	85	170	
25	81	163	
26	78	157	
27	75	151	
28	73	146	
29	70	141	
30	68	136	
31	66	132	
32	64	128	

with Option H 81 (16 K tool memory)

Basic version with 8 K tool memory

Caution!

A change only becomes active after formatting the user memory (softkey "FORMAT USER M.").

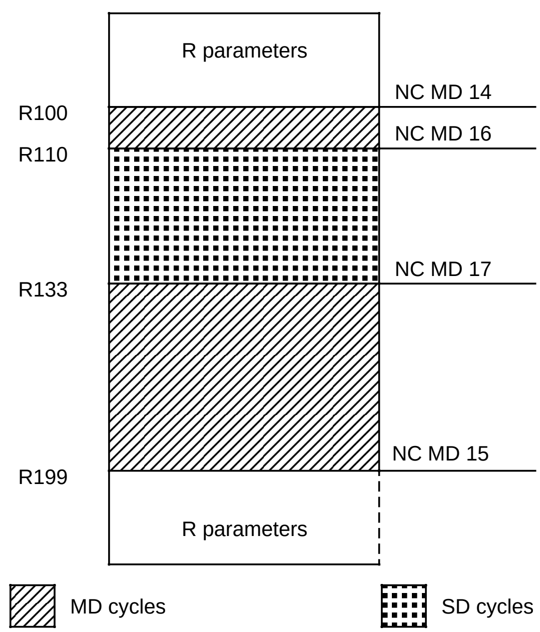
14, 15 16, 17	Measuring cycles machine data Measuring cycles setting data		14, 15 16, 17
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**999**	**-**

The measuring cycles machine data and measuring cycles setting data are located in the channel-specific R parameters. Areas for measuring cycles MD and for measuring cycles SD are specified in the R parameters for each NC machine data (NC MD 14 to 17). If these areas coincide, the corresponding R parameters for the measuring cycles setting data take priority. The measuring cycles machine data are inhibited via the password, while the measuring cycles setting data are inhibited by means of a key switch as a function of NC MD 5005 bit 3. Inhibiting with all measuring cycles data is only active from the keyboard. Since these machine data are for the general areas of the measuring cycles MD and measuring cycles SD, a measuring cycles MD/SD area in the channel-specific R parameters is identical for each channel.

The measuring cycles MD and measuring cycles SD are fully assigned by the Option B78 (measuring cycle version 10) and FB package 1 (tool management).

During standard installation the following values are entered in the NC machine data:

NC MD 14	=	100	from	password-protected R parameters
NC MD 15	=	199	to	
NC MD 16	=	110	from	key switch-protected R parameters
NC MD 17	=	133	to	



The measuring cycles (option B78) of version 20 are no longer located in R parameters R110 to R199, but the newly created cycle MDs and cycle SDs in Software Version 3. For reasons of compatibility it is still possible to define certain R parameters as measuring MDs/SDs and to protect them from unwanted input with a password or key switch.

18	**Zero offset group**		**18**
Standard value	Lower input limit	Upper input limit	Units
1	**0**	**10**	**-**

This MD has only an internal significance when using the Siemens cycle L960.

19	**Next cutting edge for grooving cycle L93**		**19**
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**10**	**-**

The grooving cycle L93 and the measuring cycles (Option B78) use two-edged tools. The second edge is assumed to be addressed with the tool offset D_{x+1} (first edge ...D_x). If, however, the Siemens tool management of SINUMERIK 880 GA2 is used (cycle MD 7000 bits 4 to 1), the tool offset number for the second edge is transferred from the tool management in an R parameter. If, for reasons of compatibility, the tool management of SINUMERIK 850 (only when upgrading from 850 to 880) or a tool management from another manufacturer is used, a P memory (in the TO memory) must be input in NC MD 19.

NC MD 19 specifies the P memory (in the TO memory) where the reference to the next cutting edge is located when multiple edge tools are used. The reference can only be located in P memories 5 to 10.

20	**Basic angle for nutating head**		**20**
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**±180 000**	**10^{-3} deg.**

For description see NC MD 5010 (5D tool length compensation).

 6ZB5 410-0HG02

21	Power of the spindle		21
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**9999**	**kW**

For an exact description and meaning see technology calculator (Option).
Without a technology calculator NC MD 21 is meaningless.

23	Number of buffer pairs for CP 231		23
Standard value	Lower input limit	Upper input limit	Units
16	**1**	**16**	**-**

Active: After POWER ON

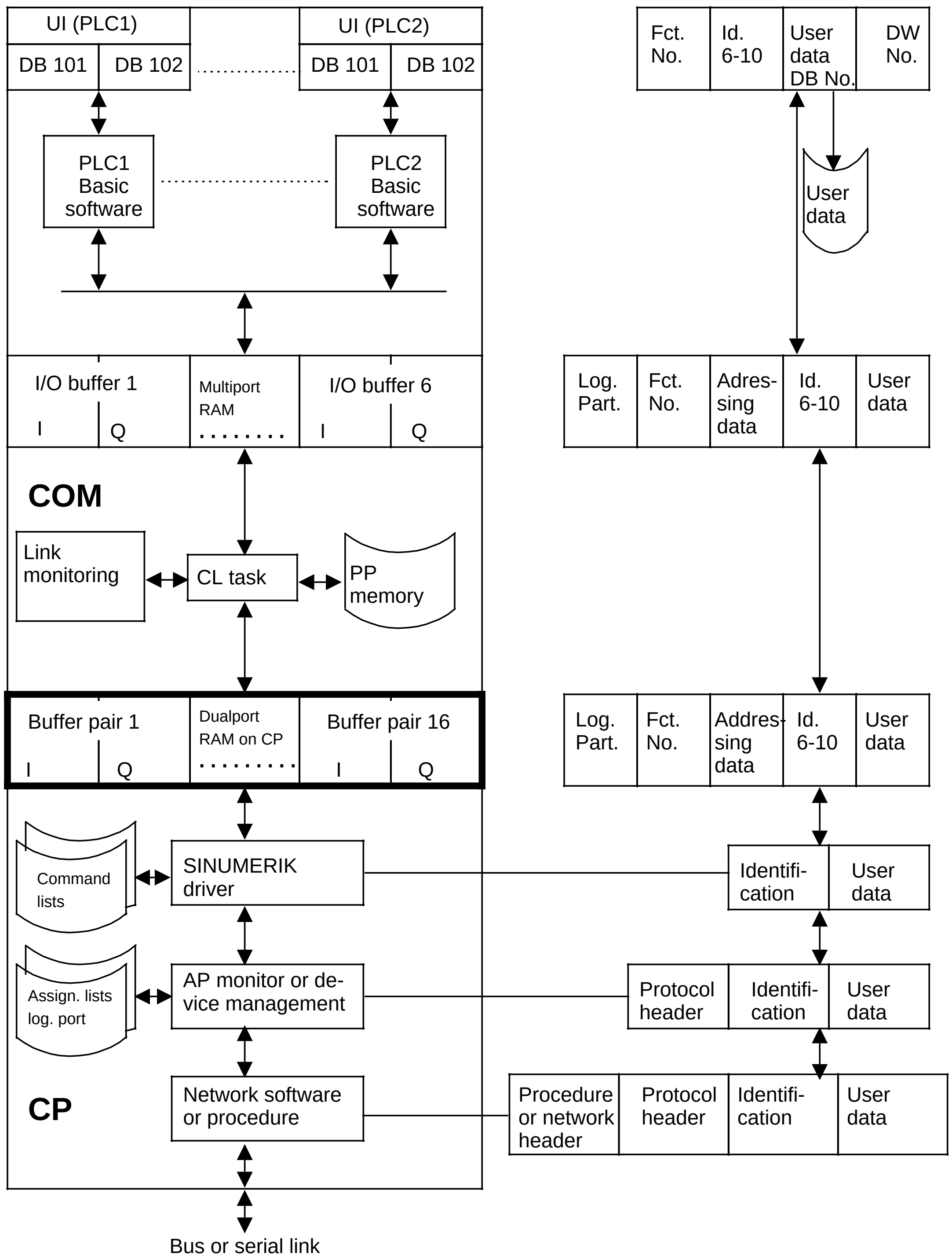
24	Number of buffer pairs for CP 315-1		24
Standard value	Lower input limit	Upper input limit	Units
16	**1**	**16**	**-**

Active: After POWER ON

For exact description see computer link.

25	Number of buffer pairs for CP 315-2		25
Standard value	Lower input limit	Upper input limit	Units
16	**1**	**16**	**-**

Active: After POWER ON


UI (PLC1)
DB 101
DB 102
UI (PLC2)
DB 101
DB 102
PLC1 Basic software
PLC2 Basic software
I/O buffer 1
I
Q
Multiport RAM
I/O buffer 6
I
Q
COM
Link monitoring
CL task
PP memory
Buffer pair 1
I
Q
Dualport RAM on CP
Buffer pair 16
I
Q
Command lists
SINUMERIK driver
Assign. lists log. port
AP monitor or device management
CP
Network software or procedure
Bus or serial link
Fct. No.
Id. 6-10
User data DB No.
DW No.
User data
Log. Part.
Fct. No.
Adres-sing data
Id. 6-10
User data
Log. Part.
Fct. No.
Adres-sing data
Id. 6-10
User data
Identifi-cation
User data
Protocol header
Identifi-cation
User data
Procedure or network header
Protocol header
Identifi-cation
User data

26	UP user data length CP 231		26
Standard value	Lower input limit	Upper input limit	Units
234	**0**	**256**	

Active: After POWER ON

Specifies the maximum user data length in the message (including 10 byte identifier).

Note:

For reasons of capatability with SINUMERIK 850, 234 bytes are used.

27	UP user data length CP 315-1		27
Standard value	Lower input limit	Upper input limit	Units
234	**0**	**256**	

Active: After POWER ON

Specifies the maximum user data length in the message (including 10 byte identifier).

Note:

For reasons of compatibility with SINUMERIK 850 234 bytes are used.

28	UP user data length CP 315-2		28
Standard value	Lower input limit	Upper input limit	Units
234	**0**	**256**	

Active: After POWER ON

Specifies the maximum user data length in the message (including 10 byte identifier).

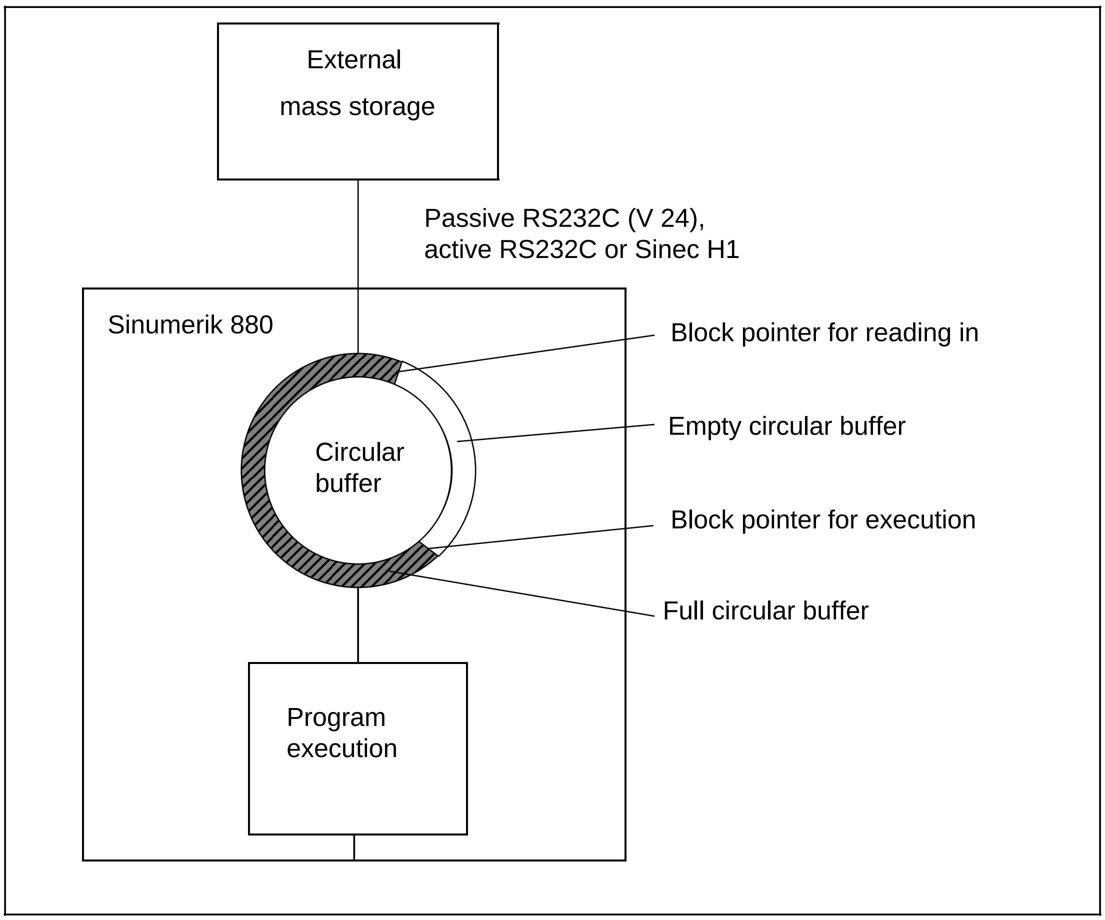
Note:

For reasons of compatibility with SINUMERIK 850 234 bytes are used.

30	Number of sectors in execution memory		30
Standard value	Lower input limit	Upper input limit	Units
10	**5**	**2 500**	**Sector**

Active: After POWER ON

During "Execution from external" (Section 11.1), the part program is read into a circular buffer while other part programs are being read out and executed. The circular buffer is part of the part program memory. The size of the circular buffer is defined in NC MD 30 by the number of sectors (1 sector = 507 bytes).



If the defined rotary buffer sectors are not available on NC START (either occupied by part programs or the part program memory is too small), the control reduces the circular buffer size to the maximum possible number of sectors.

After the function, "Execution from external" has been activated, the part program memory must be deleted.

Interface signals

Channel-specific signals to NC channel (program modification):

DB 10 - DB 25 DL2, bit 15 "Execution from external"

Channel-specific signals from NC channel (select softkey):

DB 10 - DB 25 DR14, bit 7 "Execution from external"

Read/write NC data from PLC:

FB61/ FB62

Functional description	Data type	Limit value	Value (WER 1 - WER 3)	Number format (ZFPN) FB61	Number format (ZFPN) FB62	Maximum value
Program length						
Main program length	HPLGNC	0-9999		B0, F0		
Subroutine length	UPLGNC	0-999		B0, F0		

31	Time-out for computer link		31
Standard value	Lower input limit	Upper input limit	Units
10	**0**	**9999**	**Sec.**

Active: After POWER ON

Function:

If a message is sent by the numerical control to the host computer, time-out must be switched on and the necessary response message must be received within the set time.

Behaviour:

- When transmission is triggered by the host computer:
  file transfer is returned to initial state.
- When transmission is triggered by the operator (PLC):
  file transfer is returned to initial state;
  Message to operator (error No. 3105).

Activating the function with machine data 31:

0 - Time-out function not activated.
* - Value in seconds until file transfer is aborted
  (exact value: set value - 4%).
  Standard value: 10
  Maximum value: 9999

32	Subroutine number for ISDN call number		32
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**999**	**BCD format**

Active: Immediately

This machine data specifies the subroutine containing the max. 4 ISDN numbers (call numbers). Please refer to the SINUMERIK 880 GA2 Teleservice Function Manual for the exact syntax of the subroutine.

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

Active: Immediately

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 fixed at 0%. If 0% is allocated to another switch position, the feed hold LED does not light up, unlike in position one.

Allocations of more than 150% are possible, but this value is set as the limit inside the NC.

Standard values:

1, 2, 4, 6, 8, 10, 20, 30, 40, 50, 60, 70, 75, 80, 85, 90, 95, 100, 105, 110, 115, 120;
other NC MD (NC MD 122 to 130) are set to 0.

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

Active: When all channels of mode group are in the STOP state

Assignment to max. 16 spindle override switch positions as required.

Standard values:

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

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

Active: Immediately

Standard values:
1, 10, 50, 100, 100, 100, 100, 100.

NC MD 147 1
148 10
149 50
⋮ ⋮

Rapid traverse override must be activated by means of PLC interface signals.

155	Interpolation time		155
Standard value	Lower input limit	Upper input limit	Units
8	**4**	**30**	**ms**

Active: After POWER ON

Note the following:

- Long interpolation times diminish the data throughput between the NC CPU and PLC CPU.
- $\frac{\text{MD 155 (entered value)}}{\text{MD 160 (entered value)}}$ must result in 0.5 or a whole number
- If the IPO time is divisible by 1, 2 and 4, values of 2 to 16 are allowed in MD 168 (sampling time, system clock).

- If the IPO time is divisible by 1 and 2, values of 2 to 8 are allowed in MD 168.
- If the IPO time is divisible only by 1, values of 2 to 4 are allowed in MD 168.
- With IPO times above 15 ms, the value 2 cannot be used in MD 168.

Entries that infringe the restrictions stated above are automatically corrected on Power On.

Example:

The combination

MD 155 = 14 **/** MD 160 = 1 **/** MD 168 = 16 equals 14 ms IPO, 1 ms SERVO

generates:

MD 155 = 14 **/** MD 160 = 2 **/** MD 168 = 8 equals 14 ms IPO, 1 ms SERVO

The MD 168 value should be selected as high possible in order to avoid excessively burdening the SERVO CPU with interrupt actions.

Intended to facilitate the choice of valid combinations between MD 168, MD 160* and MD 155, the enclosed table covers some of the combinations that are required in practice.

MD 155	MD 160*	MD 168	IPO [ms]	SERVO cycle time [ms]	System clock [ms]
4	2	16	4	2	1
4	2	8	4	1	0.5
4	2	4	4	0.5	0.25
5	10	4	5	2.50	0.25
5	10	2	5	1.25	0.125
6	6	8	6	3	0.5
6	3	8	6	1.5	0.5
6	3	4	6	0.75	0.25
7	7	4	7	1.75	0.25
7	7	2	7	0.875	0.125
8	4	16	8	4	1
8	2	16	8	2	1
8	1	16	8	1	1
8	1	8	8	0.5	0.5

(Continued overleaf)

MD 155	MD 160*	MD 168	IPO [ms]	SERVO cycle time [ms]	System clock [ms]
9	9	4	9	2.25	0.25
9	9	2	9	1.125	0.125
10	10	8	10	5	0.5
10	5	8	10	2.5	0.5
10	5	4	10	1.25	0.25
11	11	4	11	2.75	0.25
11	11	2	11	1.375	0.125
12	6	16	12	6	1
12	4	16	12	4	1
12	3	16	12	3	1
12	4	8	12	2	0.5
12	3	8	12	1.5	0.5
12	2	8	12	1	0.5
12	3	4	12	0.75	0.25
13	13	4	13	3.25	0.25
13	13	2	13	1.625	0.125
14	14	8	14	7	0.5
14	7	8	14	3.5	0.5
14	7	4	14	1.75	0.25
14	7	2	14	0.875	0.125
15	15	4	15	3.75	0.25
15	15	2	15	1.875	1.25
15	5	4	15	1.25	0.25
15	3	4	15	0.75	0.25
16	8	16	16	8	1
16	4	16	16	4	1
16	2	16	16	2	1
16	1	16	16	1	1
17	17	4	17	4.25	0.25

(Continued overleaf)

MD 155	MD 160*	MD 168	IPO [ms]	SERVO cycle time [ms]	System clock [ms]
18	9	8	18	4.5	0.5
18	6	8	18	3	0.5
18	9	4	18	2.25	0.25
18	3	8	18	1.5	0.5
18	3	4	18	0.75	0.25
19	19	4	19	4.75	0.25
20	5	16	20	5	1
20	4	16	20	4	1
20	2	16	20	2	1
20	1	16	20	1	1
21	7	4	21	1.75	0.25
21	3	4	21	0.75	0.25
22	11	8	21	5.5	0.5
22	11	4	21	2.25	0.25
23	23	4	23	5.75	0.25
24	4	16	24	4	1
24	3	16	24	3	1
24	2	16	24	2	1
24	3	8	24	1.5	0.5
24	1	16	24	1	1

157	**Control type for standard cycles**		**157**
Standard value	Lower input limit	Upper input limit	Units
see list	**see list**	**see list**	**-**

Active: After memory has been formatted

The identifiers in the following list are analyzed by the SIEMENS standard cycles to permit branching in the cycles for the specific control (e.g. recessing cycle with/without tool management, ...).

This MD is automatically preset with the currently installed software release and version (M/T) both during installation (load standard MD) and with every POWER ON.

SINUMERIK	Identifier	Software version	Entry in MD 157
805 T	06		
805 M	07		
810 T	11		
810 M	12		
810 G	13		
810 W	14		
820 T	21		
820 M	22		
820 G	23		
820W	24		
840 T	41		
840 M	42		
850 T/M	50		
850 T	51		
850 M	52		
880 T/M	80	6.4	8064
880 T	81	6.4	8164
880 M	82	6.4	8264
880 G	83	5.1	8351
880 N	82	4.1	8241
880 GA2 T/M	90	1.1	9011
880 GA2 T	91	1.1	9111
880 GA2 M	92	1.1	9211

158	Measuring speed		158
Standard value	Lower input limit	Upper input limit	Units
250	**0**	**12 000**	**1000 units / min (IS)**

Active: After POWER ON

When measuring in JOG mode (Option B74) pressing the softkey ”MEASURE” starts the axis. The measuring speed is entered in NC MD 158 and is valid for a feedrate override of 100%.

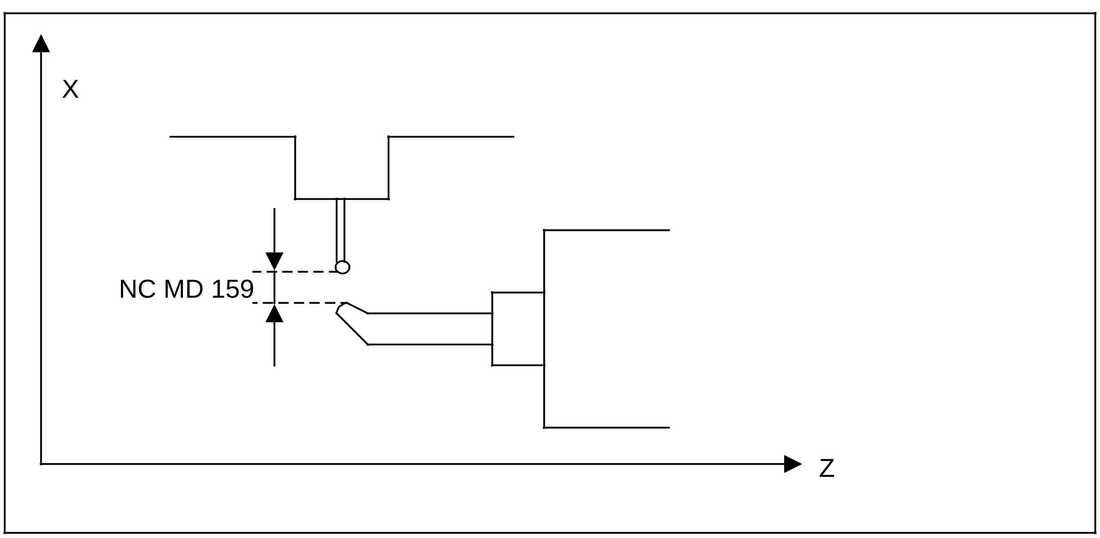
The measuring speed applies until the probe is disengaged.

159	Retraction path		159
Standard value	Lower input limit	Upper input limit	Units
10 000	**0**	**±99 999 999**	**units (IS)**

Active: After POWER ON

When measuring in JOG mode (Option B74), after the probe has been deflected it is retracted a short distance. The direction of retraction is automatically checked by the control because an incorrect direction of retraction could cause destruction of the probe. The control also checks whether manual retraction with at least the incremental path of NC MD 159 was performed. Before this no further axis movements are possible. Retraction is not possible with rapid traverse.

Example: Tool gauging



160-163	SERVO cycle time Servo CPU 1-4		160-163
Standard value	Lower input limit	Upper input limit	Units
2	**1**	**30**	**-**

Active: After POWER ON

The servo cycle time of each servo CPU is derived from the system clock (MD 168) multiplied by the value stated here.

NC-MD 160 Servo cycle time of 1st servo CPU
NC-MD 161 Servo cycle time of 2nd servo CPU
NC-MD 162 Servo cycle time of 3rd servo CPU
NC-MD 163 Servo cycle time of 4th servo CPU

Servo cycle time = System clock (NC MD 168) x input value

The standard values result in:

Servo cycle time = 1 ms x 2 = 2 ms

Condition: $$\frac{\text{Interpolation time (MD 155)}}{\text{Servo cycle time (MD 160 - 163, entered value)}} = \text{whole number or 0.5}$$

Notes:

- The position control cycle can be reduced with axis-specific effect with NC MD 1396*.
- See NC MD 155 for a table with possible input values.

164	Calculation dead time for EXTENDED THREAD PACKAGE		164
Standard value	Lower input limit	Upper input limit	Units
36	**-**	**-**	**-**

This machine data is preset by the system and must not be changed by the user. It is required for the EXTENDED THREAD PACKAGE Option.

168	Sampling interval, system clock		168
Standard value	Lower input limit	Upper input limit	Units
16	**2**	**16**	**0.0625 ms**

Active: After POWER ON

MD 168	System clock
2 4 8 16	125 µsec 250 µsec 500 µsec 1000 µsec

No other settings are allowed. The actual values are stored in the hardware with the system clock pulse.

The system clock should not, therefore, be set faster than necessary. Ideally, it is set to the servo cycle time, or to 1 msec if the servo cycle time is longer than 1 msec.

Note:

Refer to NC MD 155 as regards the input values.

172	Number of simulation tool parameters		172
Standard value	Lower input limit	Upper input limit	Units
8	**0**	**8**	**-**

With this MD, the number of dimensioning parameters for a simulation tool data record can be selected.
The more parameters are selected, the smaller the number of possible data records (constant memory area).

173	Number of simulation channels		173
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**2**	**-**

Active: After user memory has been formatted

Specifies how many simulation channels are available on the control.

SINUMERIK 880 GA2M: Input value 1
SINUMERIK 880 GA2T:
- Single slide simulation: Input value 1
- Double slide simulation: Input value 2 (NC MD 5009 bit 7 = 1)

Note:

A modification only becomes active after softkey "FORMAT USER M.".

174-176	Simulation mode of representation for plane 1 to 3		174-176
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**7**	**-**

Active: After simulation start

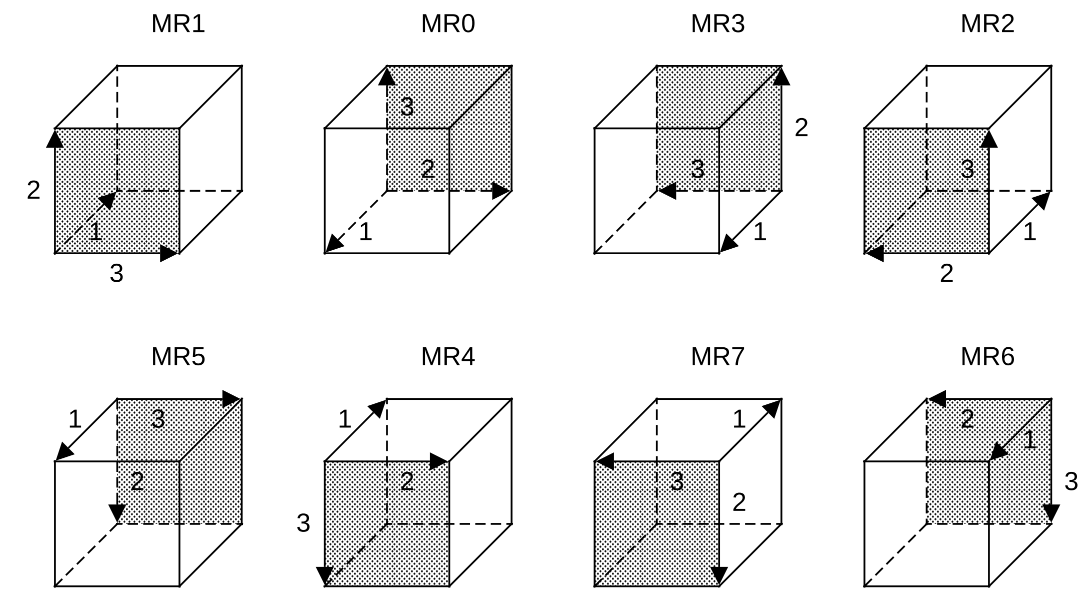
NC MD 174 Simulation mode of representation for plane 1 2nd - 3rd axis
NC MD 175 Simulation mode of representation for plane 2 1st - 3rd axis
NC MD 176 Simulation mode of representation for plane 3 1st - 2nd axis

With this machine data you can define from which direction the three-dimensional simulation process is to be seen. In this way it is possible to adapt simulation to the machine. For every simulation plane the mode of representation must be defined as shown in the following table.

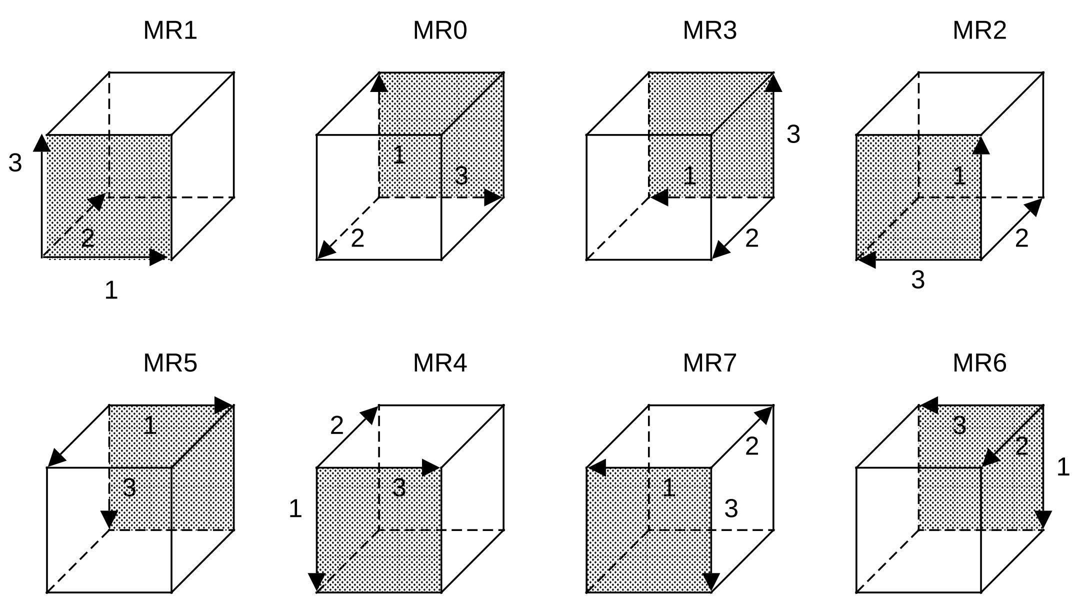
For the graphic tool check the mode of representation is taken from "mode of representation for plane 1" for the M version and from "mode of representation for plane 2" for the T version. Tools whose position is only determined in the program (cutter, drill, measuring tools) are displayed without paying attention to the mode of representation.

Mode of representation (MR) for entry in NC MD 174 to 176

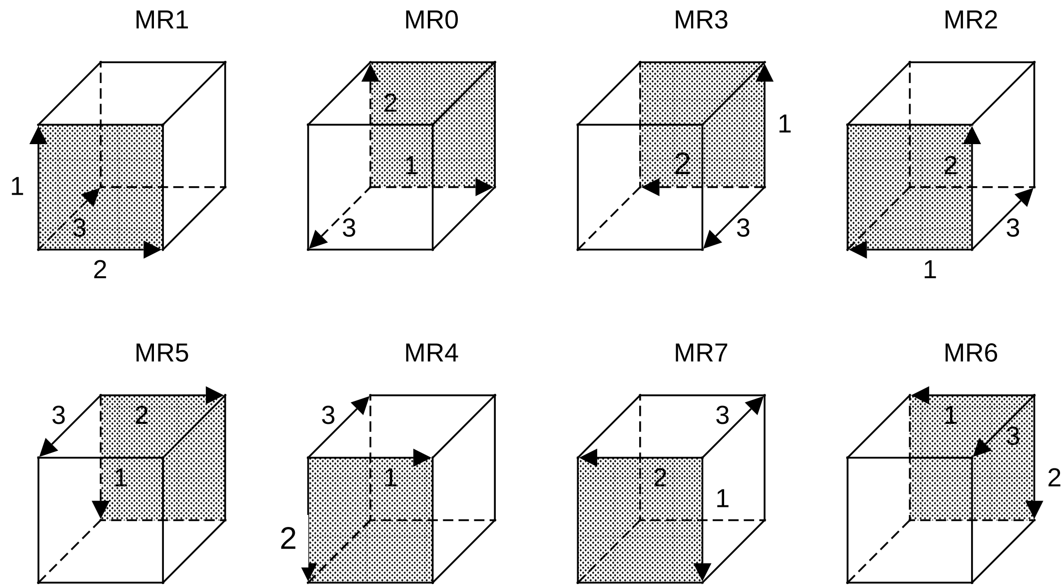
Plane 1 (M) 2nd - 3rd axis



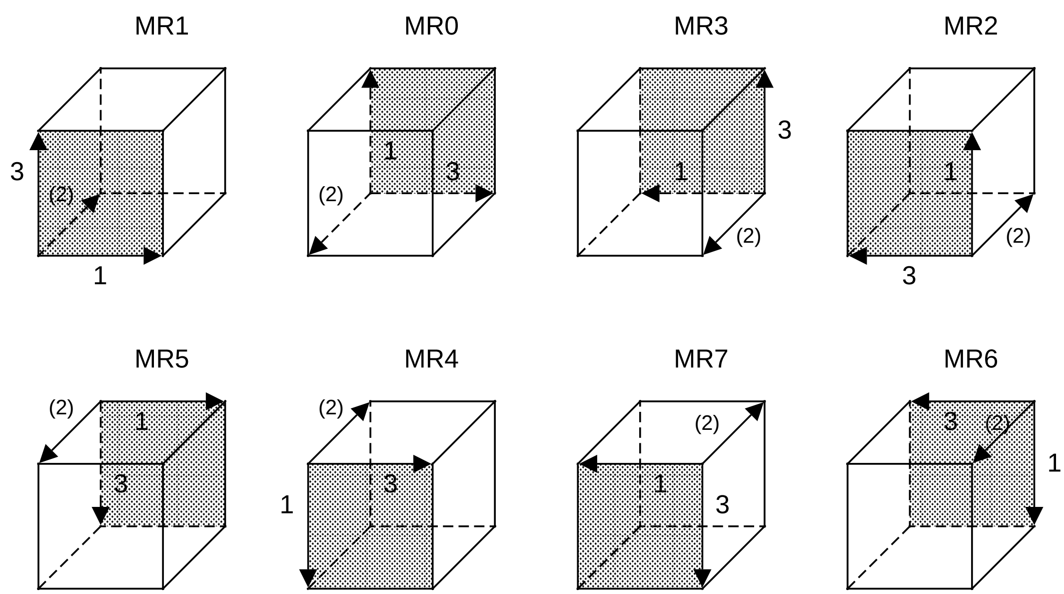
Plane 2 (M) 1st - 3rd axis



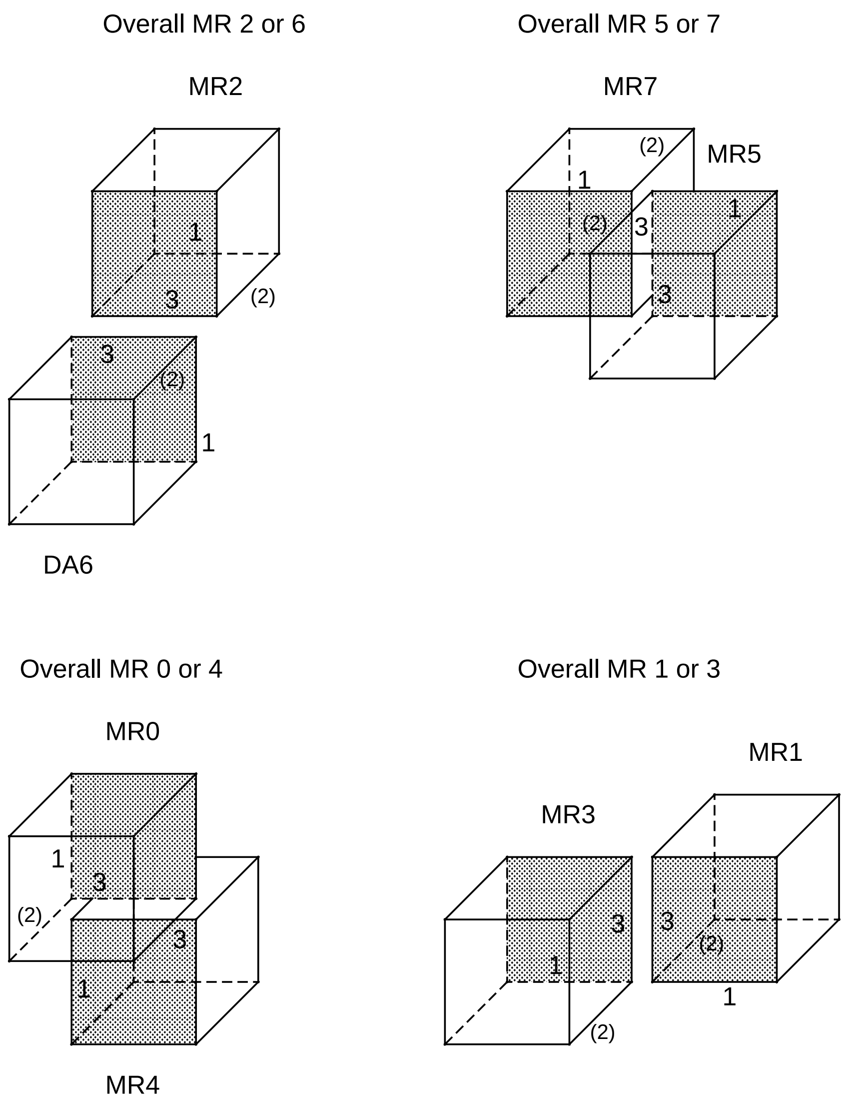
Plane 3 (M) **1st - 2nd axis**



Plane 2 (T) **1st - 3rd axis**



Plane 2 (TT)



If two slides are represented at once the modes of representation of both channels must match (see option shown above).
The mode of representation of slide 1 determines the overall mode of representation for this combination. As slides 1 and 2 can be exchanged each of the diagrams shown above shows two modes of representation.

180-199	**Coding simulation functions**		**180-199**
Standard value	Lower input limit	Upper input limit	Units
see below	**see below**	**see below**	

Active: In next block

In order to be able to perform machine-related responses in real time for graphic simulation, individual simulation functions must be adapted to the auxiliary functions defined by the manufacturer with NC MD 180 to 199.

One or two auxiliary functions can be selected per response. Each of these auxiliary functions is described by a data block consisting of two NC MD.

The following auxiliary functions can be defined:

NC MD 180 / 181	Tool change simulation channel 1	
NC MD 182 / 183	Tool change simulation channel 2	
NC MD 184 / 185	Tool offset selection	
NC MD 186 / 187	Spindle turning (1st auxiliary function)	cw
NC MD 188 / 189	Spindle turning (2nd auxiliary function)	ccw
NC MD 190 / 191	Spindle stopped (1st auxiliary function)	stop
NC MD 192 / 193	Spindle stopped (2nd auxiliary function)	in position
NC MD 194 / 195	Erase unmachined part	
NC MD 196 / 197	Draw unmachined part	
NC MD 198 / 199	Synchronization mark for double slide operation.	

Standard machine data:

NC MD	Input Value	Function	Aux. funct.
180/181	63/0	Tool change Simulation channel 1	T...
182/183	63/0	Tool change Simulation channel 2	T...
184/185	64/0	Tool offset selection	D...
186/187	21/3	Spindle turning cw	M03
188/189	21/4	Spindle turning ccw	M04
190/191	21/5	Spindle stopped stop	M05
192/193	21/19	Spindle stopped in position	M19
194/195	21/100	Erase unmachined part	M100
196/197	21/101	Draw unmachined part	M101
198/199	21/46	Synchronization mark for doubleslide operation	M46

NC MD 180			NC MD 181
0 to 99	**0 to 7**	**0 to 5**	**0 to 99 999 999**
Address extension	Response bits	Addr. of the aux. funct.	Number of the auxiliary function

M function: 0 to 999
S function: 0 to 16 000
T function: 0 to 99 999 999
D function: 0 to 409
H function: 0 to 99 999 999

0 . . . . no aux. function
1 . . . . M function
2 . . . . S function
3 . . . . T function
4 . . . . D function
5 . . . . H function

Response bits for evaluation

- of the axis letter
- of the axis address
- of the aux. funct. number

0			

Address letter is not evaluated
Address passing on is not evaluated
Aux. function number is not evaluated
always zero

The BCD value of these 4 response bits must be input

Address extension

200	PLC serial interface		200
Standard value	Lower input limit	Upper input limit	Units
3	1	4	-

Active: After POWER ON

In DB 37 the PLC is able to input and output any data via the serial interface entered in MD 200 (20 mA or RS 232) (see Interface Description Part 1). The interface on the COM CPU is preset with the standard value 3.

MD 200 can be overwritten with FB 62, provided the interface is not busy.

208	Max. tool wear [parameter P5/P6]		208
Standard value	Lower input limit	Upper input limit	Units
999 999	1	999 999	*)

Active: After POWER ON

When inputting the tool wear into length 1 (tool parameter P5) and length 2 (tool parameter P6) via the keyboard on the NC operator panel, a check is made to see whether the max. permissible value (NC MD 208) was exceeded. (Condition: NC MD 5008.3 = 1).

This prevents the tool offset (the wear!) from being unintentionally modified by a large value.

Note:

The tool wear can be declared invalid with NC MD 5007 bit 6.

209	Max. tool wear [parameter P7]		209
Standard value	Lower input limit	Upper input limit	Units
999 999	1	999 999	*)

Active: After POWER ON

Meaning as for NC MD 208 but for tool wear of the tool nose/cutter radius (tool parameter P7).

*) *1 position before the decimal point and 5 positions after the decimal point in mm/inches (NC MD 5002 bit 4).*

210	Number of TO areas		210
Standard value	Lower input limit	Upper input limit	Units
1	1	16	-

Active: After user memory has been formatted

Notes:

For pallet changers or double-slide machines, the TO memory may be subdivided into max. 16 TO areas. The advantage of such a breakdown is that the individual TO areas can be invoked with D numbers D0 to Dx, i.e. programs may be exchanged between the slides in the case of a double-slide machine since both tool turrets can be addressed with D numbers D0 to D8.

With regard to protective measures, it may also be advisable to prevent individual channels from accessing specific tool offsets.

The number of areas is specified by means of NC MD 210. The input limits are at 1 to 16 TO areas. If 0 is entered, a TO area is made available. A value exceeding 16 is limited internally to 16. At the same time, Alarm 47 is activated.

Under NC MD 211 to 226 the limits for the individual TO areas are defined with the initial TO numbers. The last TO area ranges from the last value specified under NC machine data 211 to 226 to the maximum tool offset number (observe NC MD 13).

Caution!

A change to NC MD 210 is not valid until the "FORMAT USER M." softkey has been pressed.

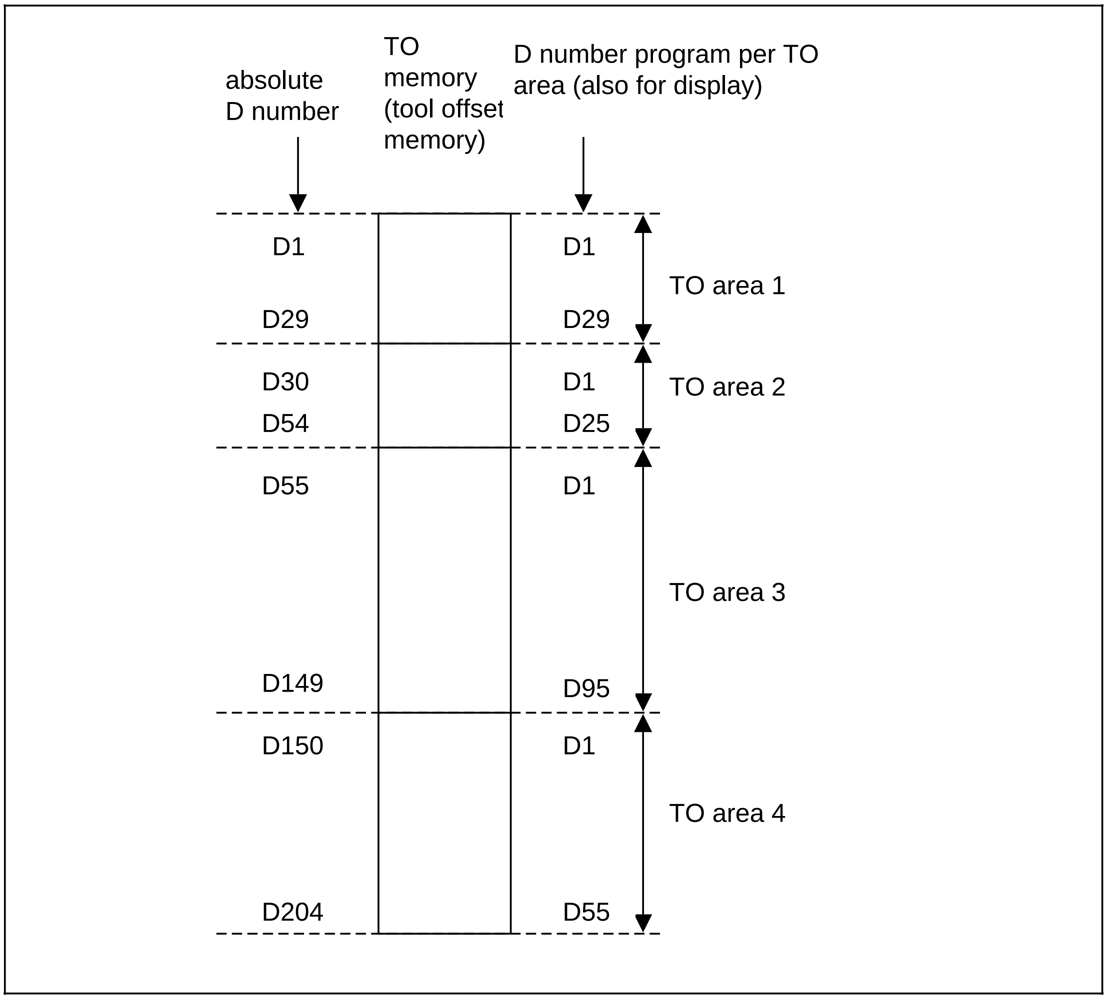
NC MD No.	Significance		Standard value
211	Initial TO number for TO area	1	1
212	Initial TO number for TO area	2	0
213	Initial TO number for TO area	3	0
214	Initial TO number for TO area	4	0
215	Initial TO number for TO area	5	0
216	Initial TO number for TO area	6	0
217	Initial TO number for TO area	7	0
218	Initial TO number for TO area	8	0
⋮	⋮		⋮
226	Initial TO number for TO area	16	0

The initial D numbers of TO areas 1 to 16 should be entered in ascending order. If not, only one TO area is set internally and Alarm 47 is activated.

Example:

NC MD	13	10	Number of TO parameters	
NC MD	210	4	Number of TO areas	
NC MD	211	1	Initial D number for TO area	1
NC MD	212	30	Initial D number for TO area	2
NC MD	213	55	Initial D number for TO area	3
NC MD	214	150	Initial D number for TO area	4
NC MD	215	0	Initial D number for TO area	5
	.		.	
	.		.	
NC MD	226	0	Initial D number for TO area	16

TO areas are assigned to the channels in NC MD 104*.



Example for the organisation of TO areas

Caution!

A change to NC MD 211 to 226 is not valid until the "FORMAT USER M." softkey has been pressed.

227	User menu for PRESET		227
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

If on selecting PRESET mode the standard system basic menu for this mode is **not** to appear on the screen, any user menu can be defined as the new basic menu for this mode by specifying the user menu number.

If the value 0 is entered the standard system basic menu appears when this mode is selected.

Note:

A modification of the NC MD only becomes valid the next time this mode is selected.

228	User menu for MDI AUTOMATIC		228
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

Meaning as for NC MD 227, but for MDI AUTOMATIC mode.

229	User menu for JOG/INC		229
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

Meaning as for NC MD 227, but for JOG or INC.

230	User menu for REPOS		230
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

Meaning as for NC MD 227, but for REPOS.

231	User menu for AUTOMATIC		231
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

Meaning as for NC MD 227, but for AUTOMATIC.

232	User menu for REFPOINT		232
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255**	**-**

Active: When this mode is next selected

Meaning as for NC MD 227, but for REFPOINT

233	No. of the PLC In the DB 38		233
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**2**	

Active: When next display is selected

With the function "file transfer operator prompt" (Computer link), you define via the MD to which PLC the data will be transferred.

234	Password LOW PG Remote		234
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**9999 9999**	**-**

235	Password HIGH PG Remote		235
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**9999 9999**	**-**

A password can be scanned for interlocking the "PG Remote" Option.

The password is contained in the general NC machine data 234 and 235. If NC machine data 234 and 235 contain a 0, no password is required (press CR when the password is requested). The programmer interprets the value entered in these machine data as an ASCII value. If NC machine data 234 contains the value 41424344 and NC machine data 235 contains the value 45464748, the password ABCDEFGH has to be entered. The entry in machine data 234 and 235 is protected by the password on the SINUMERIK 880 GA2.

251	Channel for auxiliary function of the 1st simulation channel to PLC		251
Standard value	Lower input limit	Upper input limit	Units
15	**1**	**16**	**-**

Active: After POWER ON

252	Channel for auxiliary function of the 2nd simulation channel to PLC		252
Standard value	Lower input limit	Upper input limit	Units
16	**1**	**16**	**-**

Active: After POWER ON

Auxiliary functions of the simulation channel can be output to the PLC during graphic simulation (e.g. T function for flexible tool management).
As the NC PLC interface is structured channel-specifically (8 (out16) NC channels) but no channels are provided for graphic simulation, the user can select via machine data which simulation channel he will assign to which NC channel:

Note:

Output channels assigned to NC processing (NC PLC interface) must not be used to output simulation functions.

The user can select 8 channels for NC processing and therefore 8 channel DBs as well. He can then select 2 of the remaining 8 channel DBs for outputting the auxiliary functions of the simulation channels.

The form in which the auxiliary functions are to be read out is determined in machine data 540*, 544* and 546*. The procedure here is the same as that of the NC.

The simulation program can be halted in the PLC user program with auxiliary function blocks with read-in disable, for example, to be able to declare new tools from the PLC with a programmed D No.

260	No. of M function for selecting C axis mode		260
Standard value	Lower input limit	Upper input limit	Units
–1	**–1**	**9 999**	**–**

261	No. of M function for deselecting C axis mode		261
Standard value	Lower input limit	Upper input limit	Units
–1	**–1**	**9 999**	**–**

Active: When all channels of mode group are in the STOP state

The C axis mode is selected and deselected by means of M functions. The numbers of the functions can be issued for specific customers.

When issuing the numbers, ensure that they do not conflict with other M functions. Generally speaking, the following values should not be used:

0, 1, 2, 3, 4, 5, 17, 19, 30, 36, 37

Use the address extension as for spindles when programming.

Example

MD 260=123
MD 261=456

M1=123 L_F Selection of C axis mode for spindle 1
M3=456 L_F Deselection of C axis mode for spindle 3

271-279	Cycle No. for L81/G81 bis L89/G89		271-279
Standard value	Lower input limit	Upper input limit	Units
see below	**0**	**999**	**-**

Active: In the next block

These machine data stipulate which drilling cycle is actually called when G203 or G204 and L81 to L89 is programmed.

The standard drilling cycle (L81 to L89) is called if the value is 0.

Standard values:

MD 274 = 841, all others are 0.

MD 274 is required for the "Tapping without compensating chuck" Option (also see Section 11.15).

730-739	1st transformation, parameter 1 to 10		730-739
Standard value	Lower input limit	Upper input limit	Units
0	**-99 999 999**	**99 999 999**	**units (IS)**

Active: After POWER ON

NC MD 740 to 749	2nd transformation, parameter 1 to 10
NC MD 750 to 759	3rd transformation, parameter 1 to 10
NC MD 760 to 769	4th transformation, parameter 1 to 10
NC MD 770 to 779	5th transformation, parameter 1 to 10
NC MD 780 to 789	6th transformation, parameter 1 to 10
NC MD 790 to 799	7th transformation, parameter 1 to 10
NC MD 800 to 809	8th transformation, parameter 1 to 10

The transformation parameters are required for the 2D/3D coordinate transformation.
For description see Section 11.

876-899	Leading axis coupled motion axis		876-899
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	**-**

Active: After POWER ON

With the definition of these coupled axis pairings you define which axis will be coupled with if a leading axis is programmed.

There are up to 12 coupled axis pairings.

	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis
NC MD	883	882	881	880	879	878	877	876
NC MD	891	890	889	888	887	886	885	884
NC MD	899	898	897	896	895	894	893	892

The axis number must be entered in the MD as follows:

0 . . . . . . no axis
1 . . . . . . 1st axis (NC MD 5640, bit 7 = 1)
2 . . . . . . 2nd axis (NC MD 5641, bit 7 = 1)
.
.
.
11 . . . . . 11th axis (NC MD 5650 bit 7 = 1)
. .
. .
. .

Only in the coupled axis combinations (NC MD 5156 to 5182) do you define how the coupled axis pairings will behave (coupled motion in same direction/opposite direction/...) and with which G function they will be activated.

The following must be observed for coupled axis pairings (in case of error Alarm 84 is output):

a. The axes of a coupled axis pairing must belong to the same mode group.

b. The axes of a coupled axis pairing must have the same position control resolution and the same display resolution.

c. The axes of a coupled axis pairing must belong to the same type of axis. All the following bits must concur even if they are naot relevant (e. g. "facing axis" is not relevant if "rotary aixs" = 0).

Rotary axis . . . . : MD 564*, 5
Actual value display modulo 360 degrees . : MD 560*, 7
Modulo progr. for rotary axes . . . . : MD 572*, 2
Rounding axis . . . . : MD 560*, 3
Rounding up to whole/half degree . . . . . . : MD 560*, 2
Auxiliary axis . . . . : MD 572*, 0
Facing axis . . . . : MD 572*, 1

d. The axes of a coupled axis pairing must be able to be programmed in the same channels MD 576*, bits 7 to 0 and MD 580*, bits 7 to 0.

e. The axes of a coupled axis pairing must be available (NC MD 564*, bit 7 = 1).

f. Neither axis of a coupled axis pairing can be a fictitious axis (NC MD 564*, bit 6 = 0).

g. One leading axis can be coupled to several axes.

Example:

	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis
NC MD	883	882	881	880	879	878	877	876
	0	**0**	**6**	**1**	**5**	**1**	**4**	**1**

h. The leading axis of a coupled axis pairing can **not** at the same time be the coupled axis axis of the same pairing (also observe NC MD 5156 to 5188).

	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis
NC MD	883	882	881	880	879	878	877	876
	0	**0**	**0**	**0**	**3**	**2**	**2**	**1**

not possible

i. The leading axis of a coupled axis pairing can at the same time be the coupled motion axis of another pairing (observe also NC MD 5156 to 5183).

j. A leading axis can **not** be coupled to itself.

	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis	Coupled axis	Leading axis
NC MD	883	882	881	880	879	878	877	876
	0	**0**	**0**	**0**	**0**	**0**	**1**	**1**

k. A coupled axis can be coupled to several leading axes, if the leading axes are in different coupled motion pairings and these coupled axis pairings are not active at the same time.

The definition of the coupled axis pairings can be changed with a warm restart without POWER ON (for warm restart also see Section 11.6).

Note:

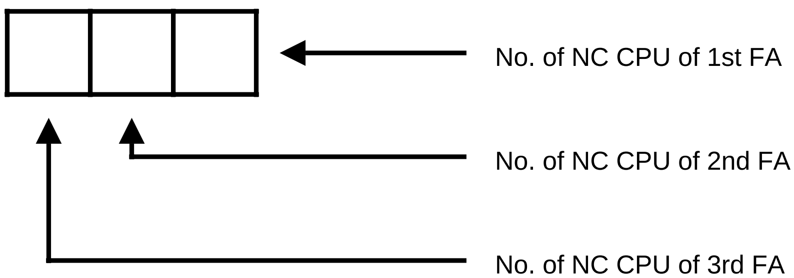
The "Coupled motion of axes" function is an Option.

930	Assignment of FA to NC CPU		930
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**222**	**–**

Active: After warm restart

The information entered in this machine data identifies which NC CPU will be used for calculating the particular electronic gearbox (ELG) grouping. The assignments of all 3 following axes (FA) are defined with NC MD 930.

Structure of MD 930:



Examples:

0 1 1 Calculation of FA1 and FA2 occurs on NC CPU I
0 2 1 Calculation of FA1 occurs on NC CPU I, of FA02 on NC CPU II

This machine data becomes effective with **warm start**.

931	Differential measurement		931
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**2**	**–**

Active: Immediately

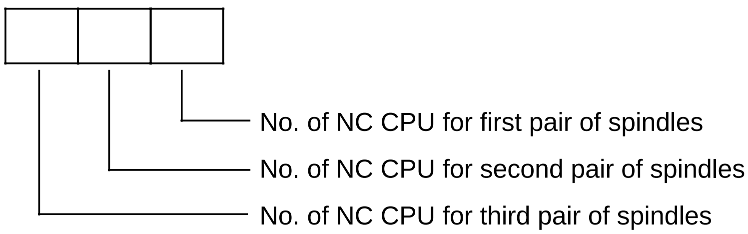
With the "Tapping without compensating chuck" function, a differential measurement that indicates the thread error can be activated with the installation routine. For this purpose the value "2" has to be entered in NC MD 931.

For a more detailed explanation, refer to the functional description "Tapping without compensating chuck" in Section 11.15.

932	Assignment of synchronous spindles to NC CPUs		932
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**222**	**–**

The information entered in this machine data identifies which NC CPU will perform the calculations for the individual pairs of synchronous spindles.

The structure of MD 932 is as follows:



Notes

- In MD 930 the ELG pairs are assigned similarly to the NC CPUs.
- The sum of the pairs of synchronous spindles and ELG groupings must not be more than three. Therefore, the individual boxes in MD 930 and 932 may only be used in one of the two machine data with a value not equal to zero.
- The given NC CPU must process the master channel of the mode group of the spindles.
- When making the entry, remember that the numbers of the NC CPUs are not checked.

940/ 960/980	Global C axis No. of following spindles 1/2/3 (synchronous spindles)		940/ 960/980
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	**–**

Each following spindle must be assigned a C axis in MD 461*. As in MD 461*, the global axis No. of this C axis must be entered here too.

940/ 960/980	**Global axis number of following axes 1/2/3 (ELG)**		**940/ 960/980**
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	

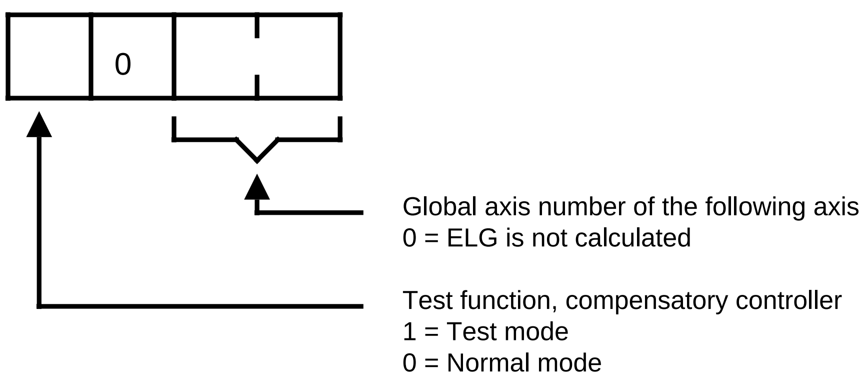
Active: After warm restart

The leading and following axes of the ELG groupings must be configured. To do this, the global axis numbers of the following axes and the associated leading axes are entered in the appropriate machine data. All following and leading axes have the standard default "0" (no axis).

The global axis number of the following axis must be entered as positive integer. If "0" is entered, this ELG is not calculated.

For setting the compensatory controller, it is possible to activate a built-in test function with this MD.

The structure of the MD:



With this test function, a disturbance on the leading axis can be simulated. The test mode is switched on and off with **warm restart**.

ELG MD

941-942	Global axis number of the first and second fictitious leading axis for following axis 1		941-942
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	

961-962	Global axis number of the first and second fictitious leading axis for following axis 2		961-962
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	

981-982	Global axis number of the first and second ficticious leading axis for following axis 3		981-982
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	

Active: After warm restart

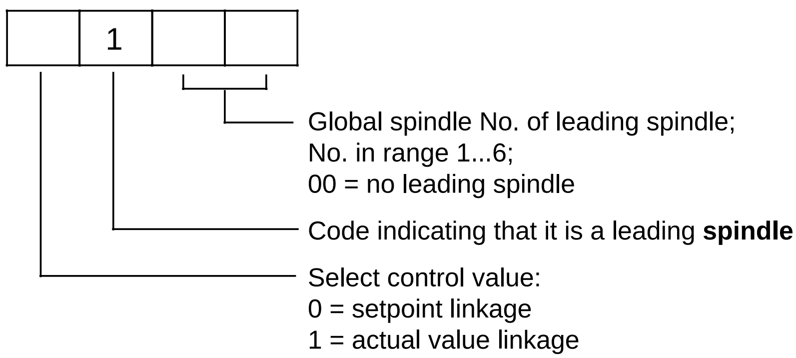
The global axis numbers for the fictitious leading axes must be entered as positive integers. If the axis number "0" is entered for a fictitious leading axis, this leading axis is considered to be not available.

The MD values become active with **warm restart**.

943/ 963/983	Configuration of leading spindles to following spindles 1/2/3		943/ 963/983
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 106**	**Special format**

The global spindle numbers of the leading spindles of the individual pairs of synchronous spindles are entered in this machine data. Also established is whether the setpoint or the actual value of the leading spindle is to be used as the control value for the following spindle.

The structure of the machine data is as follows:



Notes

- The two spindles of a pair of synchronous spindles must be calculated on the same NC CPU.
- One spindle may not be a following spindle in two different pairs of synchronous spindles.
- A following spindle may not be declared as the leading spindle of another pair (i.e. no cascading possible).
- The position-control sampling intervals of the leading spindle and following spindle must be identical in the following cases:
  - for setpoint linkage
  - for a defined value of angular offset between leading spindle and following spindle.

Examples

- First spindle as leading spindle in first pair of synchronous spindles with setpoint linkage:
  MD 943 = 0101
- Third spindle as leading spindle in second pair of synchronous spindles with actual value linkage:
  MD 963 = 1103

ELG MD

943/ 944/945	Global axis number of the first, second and third real leading axis for following axis 1		943/ 944/945
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 124**	

963/ 964/965	Global axis number of the first, second and third real leading axis for following axis 2		963/ 964/965
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 124**	

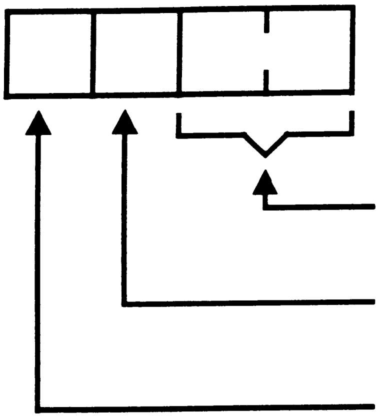
983/ 984/985	Global axis number of the first, second and third real leading axis for following axis 3		983/ 984/985
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 124**	

Active: After warm restart

The global axis numbers of the 1st/2nd/3rd real leading axes must be entered as positive integers. If "0" is entered for an axis number, this leading axis is not considered for setpoint calculation of the following axis. In addition, the information as to whether the leading axis is an axis or a spindle and whether the setpoint value (only possible for axis) or the actual value is to be used as the command variable.

The MD values become active with **warm restart**.

Structure of the MD for definition of the real leading axes:



Global axis/spindle number (2 BCD places) of the LA
00 = Leading axis not available

"Spindle bit": 0 = Axis is LA
1 = Spindle is LA

Setpoint/actual value linkage:
0 = LA automatically controlled (setpoint linkage)
1 = LA not automatically controlled (actual value linkage)
(when LA = spindle, only 1 is possible)

Examples for entry:

3rd axis as real loading axis with setpoint linkage (standard case)
Enter 0003 or 3

2nd spindle as real leading axis with actual value linkage
Enter 1102

When configuring the ELG groupings (definition of leading and following axes) the following must be observed:

- All axes in an ELG grouping (LA and FA) must belong to the same mode group. Programming is possible in different NC channels.
- All axes in an ELG grouping (LA and FA) must be calculated on the same NC CPU.
- A following axis must not be declared as leading axis for another following axis (cascading is not possible).
- A leading axis can be assigned to several following axes.
- The ELG groupings can be reconfigured via the machine data. With "warm restart", the new ELG groupings are transferred.
- All axes in an ELG grouping must be available (NC MD 564*, bit 6 = 0).
- The axes in an ELG grouping can be of different type (round axis, linear axis MD 564*, bit 5) or have different display resolution (MD 1800*, bits 4-7).
- The position control sampling times of leading and following axes must be identical in the following cases:
  - with setpoint linkage
  - when applying on-the-fly synchronization between leading and following axis (both for actual value and setpoint linkage).
- A leading axis can be switched in the follow-up mode. To make sure that the hardware monitoring of the actual value encoder is active, a setpoint output must be assigned to this leading axis.
- For an overlaying movement (FA overlay, on-the-fly synchronization, semi-automatic centering), the following axis requires a free NC channel.

946/ 966/986	Control par. for compensatory controller of following spindles 1/2/3 (P-component)		946/ 966/986
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

947/ 967/987	Control par. for compensatory controller of following spindles 1/2/3 (I-component)		947/ 967/987
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

948/ 968/988	Control par. for compensatory controller of following spindles 1/2/3 (D-component)		948/ 968/988
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

The control behaviour of the PI-action compensatory controller is set with these machine data.

Adjustment instructions for the compensatory controller
with synchronous spindles are given in

SINUMERIK 880, Function Manual,
Extended Spindle Functions

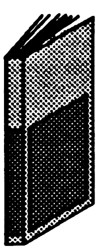
ELG MD

946/ 947/948	Control parameters for compensat. controller of the following axis 1 (P, I, D components)		946/ 947/948
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

966/ 967/968	Control parameters for compensat. controller of the following axis 2 (P, I, D components)		966/ 967/968
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

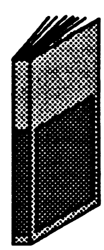
986/ 987/988	Control parameters for compensat. controller of the following axis 3 (P, I, D components)		986/ 987/988
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.001

The control behaviour of the PID-action compensatory controller is set with these machine data.



Adjustment instructions for teh compensatory controller
with the eletronic gearbox are given in

SINUMERIK 880, Function Manual,
Electronic Gearbox



950/ 970/990	Time constant parallel model for following axis/spindles 1/2/3		950/ 970/990
Standard value	Lower input limit	Upper input limit	Units
0	0	16 000	0.01 ms

Active: After warm restart

If the value 16000 is entered in the machine data, the effective Kv factor and the time constant of the following axis/spindle are automatically calculated by the servo. This effective K_V factor is simply stored internally, however, and cannot be consulted.

The value 16000 entered for the time constant is also automatically replaced by the internally calculated value.

Further notes:

See the Function Manuals "Electronic Gearbox" and "Extended Spindle Functions".

ELG MD

952-953	Tolerance band for synchronism fine and coarse of the following axis 1		952-953
Standard value	Lower input limit	Upper input limit	Units
fine 40 coarse 100	0	16 000	units (MS)

972-973	Tolerance band for synchronism fine and coarse of the following axis 2		972-973
Standard value	Lower input limit	Upper input limit	Units
fine 40 coarse 100	0	16 000	units (MS)

992-993	Tolerance band for synchronism fine and coarse of the following axis 3		992-993
Standard value	Lower input limit	Upper input limit	Units
fine 40 coarse 100	0	16 000	units (MS)

Active: After warm restart

During LINK ACTIVE, the positional deviation of the following axis to the leading axes is monitored with the tolerance band "Synchronism fine" and "Synchronism coarse".

If the positional deviation FA/LA is greater than the tolerance band, the corresponding PLC interface signal "Synchronism fine" or "Synchronism coarse" is set to 0 signal.

With the message "Synchronism fine" or "Synchronism coarse", the positional synchronism of the following axis can thus be tested.

952/ 972/992	Fine synchronism for following spindles 1/2/3		952/ 972/992
Standard value	Lower input limit	Upper input limit	Units
40	0	16 000	Units (MS)

953/ 973/993	Coarse synchronism for following spindles 1/2/3		953/ 973/993
Standard value	Lower input limit	Upper input limit	Units
100	0	16 000	Units (MS)

The positional deviation between leading spindle and following spindle is monitored in the synchronous mode. It is performed with the two tolerance bands "Fine synchronism" and "Coarse synchronism".

For each tolerance band there is an interface signal that is set when the position difference between leading spindle and following spindle fluctuates within the band.

This allows the positional synchronism between leading spindle and following spindle to be interrogated by the PLC user program by means of IS: FINE SYNCHRONISM and COARSE SYNCHRONISM.

Notes

- Any changes to these values become operative immediately.
- For an assigned C axis the clamping tolerance (NC ND 212*) must be greater than the exact stop window "coarse synchronism".

ELG MD

954/ 974/994	Emergency retraction threshold for following axes 1/2/3		954/ 974/994
Standard value	Lower input limit	Upper input limit	Units
400	0	16 000	units (MS)

Active: Immediately

With LINK ACTIVE the positional deviation of the following axis to the leading axes can be monitored with the machine data "Emergency retraction threshold".

If the positional deviation FA/LA exceeds this threshold value, it can very quickly be output by means of a digital hardware signal. In addition, an NC alarm ("FA emergency retraction") and a PLC interface signal "EMERGENCY retraction active" are set.

For the very fast message "EMERGENCY retraction" (in the position control cycle) the mixed peripherals module (N79) must be used in the servo area.

Changes to the machine data value become active immediately.

ELG MD

955/ 975/995	% warning threshold n_{max} and a_{max} for following axes 1/2/3		955/ 975/995
Standard value	Lower input limit	Upper input limit	Units
90	0	100	%

Active: After warm restart

The following axis is limited to maximum acceleration and maximum velocity (MD 276*, 280*).

In addition, in both cases a warning threshold is checked whose value is defined as a percentage of the maximum value in MD 9*5. The percentage value applies for both limits.

If, for example, 50 is entered in MD 276* as acceleration, a PLC interface signal is output in the standard setting if the value of 45 is exceeded.

If the calculated set velocity/set acceleration of the following axis is greater than the specified values, the associated interface signals are set at the PLC interface.

954/ 974/994	Emergency retraction threshold for following spindles 1/2/3		954/ 974/994
Standard value	Lower input limit	Upper input limit	Units
400	0	16 000	Units (MS)

The positional deviation between leading spindle and following spindle is monitored in the synchronous mode. In addition to the two tolerance bands for "Fine synchronism" and "Coarse synchronism" (MD 952/972/992 and MD 953/973/993) there is also emergency retraction monitoring with the threshold value "Emergency retraction threshold".

If the positional deviation exceeds this threshold value, the "Emergency retraction" alarm is triggered and IS: EMERGENCY RETRACTION ACTIVE set. However, this will only happen if the emergency retraction monitoring has been activated (SD bit "Enable emergency retraction" set).

When the mixed I/O module in the servo area is used, the "Emergency retraction" signal can also be output as a digital signal (see section on "Hardware signal - emergency retraction"). This is much faster (at position controller clock pulse speed) than evaluating the interface signal.

Note

Any changes to the these values become operative immediately.

955/ 975/995	Speed and acceleration warning thresholds for following spindles 1/2/3		955/ 975/995
Standard value	Lower input limit	Upper input limit	Units
90	0	100	%

Warning thresholds for the speed and acceleration of the following spindle are established with these machine data.

The input values describe the percentages of the actual limit values for speed and acceleration (depending on the actual gear ratio, spindle operating mode, etc.). The limit values are determined by the minima of the relevant values for leading spindle and following spindle.

If these warning thresholds are exceeded, IS: SPEED WARNING THRESHOLD CROSSED and ACCELERATION WARNING THRESHOLD CROSSED are set. The PLC user program can then reduce the programmed speed accordingly.

ELG MD

956/ 976/996	Waiting time automatically controlled correction for following axes 1/2/3		956/ 976/996
Standard value	Lower input limit	Upper input limit	Units
16 000	**0**	**16 000**	**1 ms**

Active: After warm restart

If disturbances occur in the leading axes, the following axis changes over to automatically controlled correction, i.e. travel with actual values as the command variable. When the waiting time specified above has expired, the system switches over from "automatically controlled" correction to "normal correction" (FA in follow-up mode).

Effect of the input values (case distinction):

0 : No automatically controlled correction : immediately normal correction
1 ... 15000 : First automatically controlled correction: after expiry of the waiting time, switchover to normal follow-up mode

15001 and greater : Permanent automatically controlled correction; no switching over to normal follow-up mode

956/ 976/996	Automatically controlled correction delay for following spindles 1/2/3		956/ 976/996
Standard value	Lower input limit	Upper input limit	Units
16 000	**0**	**16 000**	**ms**

If a fault such as triggering of the zero speed control of an axis stops the mode group to which the leading spindle is assigned, the leading spindle will go into follow-up mode and the following spindle will go initially into "automatically controlled correction".

In this status only the actual values of the leading spindle are then used as for control values the following spindle (even when parameters have been assigned for setpoint linkage).

After the preset delay, the following spindle is changed over from automatically controlled correction to the normal follow-up mode in which there is a rapid deceleration at maximum deceleration current. Then only the actual values of position are held and the linkage between leading spindle and following spindle is no longer maintained.

There are three different cases for entering delay time values:

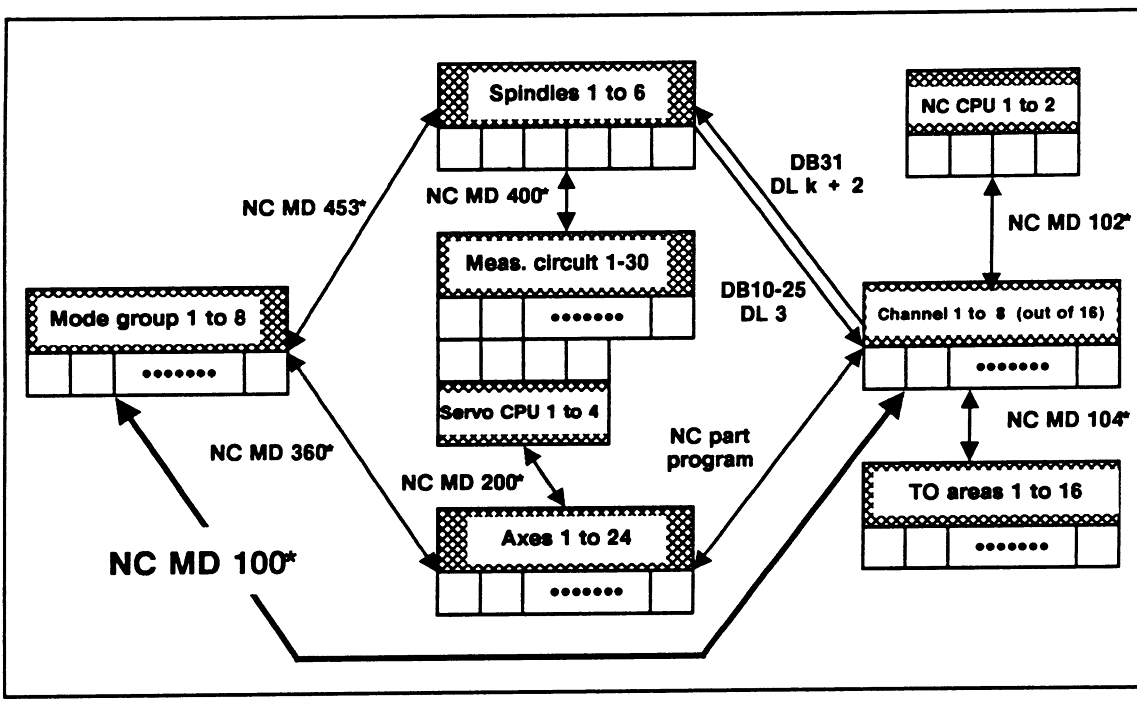
- Value 0: No automatically controlled correction; immediate changeover to normal follow-up mode in the event of leading spindle malfunction.
- Values 1...15 000: Automatically controlled correction initially; changeover to normal follow-up mode after delay.
- Values over 15 000: Automatically controlled correction only; no changeover to normal follow-up mode.

9.3.5.2 Channel-specific values

100*	Channel valid in operating mode group		100*
Standard value	Lower input limit	Upper input limit	Units
Channel No. 1, 2, 3 = 1 Channel No. > 3 = 0	0	8	-

Active: After warm restart

In the SINUMERIK 880 GA2 up to eight mode groups can be selected. All channels are assigned to the corresponding mode groups by means of NC-MD 100*. At least 1 channel and no more than 8 channels may be assigned to a mode group. The assignment can be altered with a warm restart without POWER ON reset (also see Section 11.6 for warm restart). Channels do not have to be activated in ascending order.



102*	Number of corresponding NC CPU		102*
Standard value	Lower input limit	Upper input limit	Units
Channel No. 1, 2, 3 = 1 **Channel No. > 3 = 0**	**0**	**2**	**-**

Active: After POWER ON

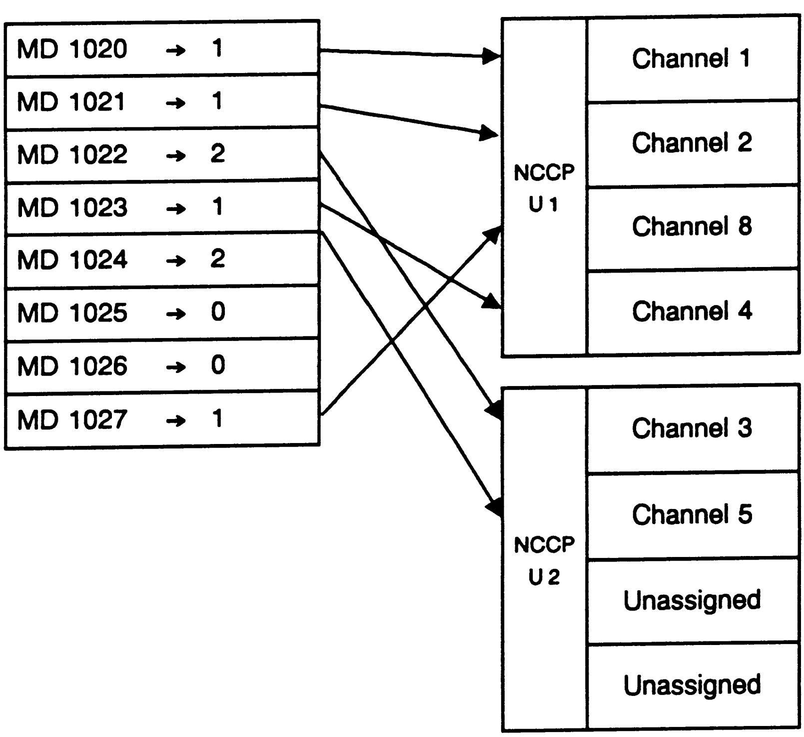
No more than two NC CPUs may be used with the SINUMERIK 880 GA2 (the maximum number depends on the version). All available channels are assigned to the corresponding NC CPU by means of NC MD 102*. A channel is deemed to be available if it has been assigned to a mode group (MD 100* = 0 means no assignment). Up to 4 channels may be assigned to one NC CPU.

All activated channels can be distributed to the availabe NC CPUs in any order. However, at least **one channel** must be available on every NC CPU.

The number of channels activated on one NC CPU should only be that which is actually used, otherwise processor power is wasted (block preparation time can increase).

For the maximum number of channels per NC CPU also see Section 9.1.2.

Typical channel assignment:



104*	TO area for channel		104*
Standard value	Lower input limit	Upper input limit	Units
1	1	16	-

Active: After user memory has been formatted

The TO area for each available channel is entered in this MD. The maximum value is to be selected in accordance with NC MD 210 (Also see MD 210, 211-226).

106*	Number of released program		106*
Standard value	Lower input limit	Upper input limit	Units
0	see below	see below	-

Active: In next block

When 0 is entered, all programs at the first processing level are released for this channel. If only a specific program at the first processing level can be activated in a given channel, the program number should be entered at this point.

This can be useful if only one particular movement is to be carried out in a channel, e.g. for loaders, tool palleting, ...

Main programs:	1	to	+9999
Subroutine/cycles:	–1	to	–999
All main programs and subroutines:	0		

108*-120*	Reset position for G groups		108*-120*
Standard value	Lower input limit	Upper input limit	Units
see below	see below	see below	-

The reset position for the G groups 1, 3, 6, 8, 12 and 15 can now be defined channel-specific.

Only the G functions of the corresponding G group listed in the table below can be entered in the NC MD 108* to 120*. The reset position for G group 9 (G70/G71) is defined in NC MD 5002 with the input resolution.

Standard value	NC MD for reset pos.	G-group													
G01	108*	1	G13	G12	G06	G37	G35	G34	G33	G03	G02	G11	G10	M/T G01	G00
		2													G09
M: G17 T: G18	110*	3										G09	G19	T G18	M G17
		4											G42	G41	M/T G40
		5													G53
G54	112*	6										G57	G56	G55	M/T G54
		7							G74	G92	G59	G58	G26	G25	G04
M: G60 T: G64	114*	8										G62	T G64	G63	T G60
		9												G71	G70
		10				G89	G88	G87	G86	G85	G84	G83	G82	G81	M/T G80
		11											G68	G91	M/T G90
M: G94 T: G95	118*	12										G97	G96	T G95	M G94
		13					G111	G110	G48	G348	G248	G148	G347	G247	G147
		14												G51	M/T G50
G150	120*	15				G159	G158	G157	G156	G155	G154	G153	G152	G151	M/T G150
		16										G135	G133	G131	G130
		17										G235	G233	G231	G230
		18										G335	G333	G331	G330

For G group allocation and the description of the G functions see Programming Guide.

130*	Serial interface for "Execution from external"		130*
Standard value	Lower input limit	Upper input limit	Units
3	0	4	-

This machine data is used to select the interface (RS232C, file transfer) which is to be started for the channel in question for the "Execution from external" function.

Channel No. x = 0 → Channel 1
x = 1 → Channel 2
etc.

Interface 0: no interface
Interface 1-4: 1st - 4th RS232C (V.24) interface
Interface 5: file transfer

Note:

The serial interface 3 (COM CPU) is used for programming the PLC with the programmer.

Using the 1st and 2nd serial interface for "Execution from external" is not recommended because the net data rate is lower than the 3rd and 4th serial interface used for computer link/file transfer.

Additional machine data: (MD 30, MD 5148-5152, option "Execution from external").

9.3.5.3 Axis-specific values / spindle-specific values

Up to 40 axes are possible and envisaged in terms of software to keep the NC system software universal and to be prepared for future developments. However, with the SINUMERIK 880 GA2, only the first 24 axes can be activated and implemented in terms of hardware (real and fictitious axes).

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

NC MD	2040	Coarse exact positioning for	1st axis
	2041	Coarse exact positioning for	2nd axis
	.	.	.
	.	.	.
	.	.	.
	.	.	.
	.	.	.
	2063	Coarse exact positioning for	24th axis
	2064	Coarse exact positioning for	25th to 40th axes
	to	(not relevant to SINUMERIK 880 GA2)	
	2079		
	2080	Fine exact positioning for	1st axis
	.		
	.	etc.	

The last position of the NC MD number is represented by the symbol "*" for the sake of clarity (e.g. 204* = coarse exact positioning). When the NC MD are displayed and input, the actual figure must be inserted for this symbol "*" in accordance with axis, spindle or channel.

200*	Axis assignment actual value input		200*
Standard value	Lower input limit	Upper input limit	Units
see below	**0**	**10 03 04 00**	**-**

Active: After POWER ON

On machines that can be ordered as modular systems, it must be possible to flexibly assign the axes actually available on the machine to the measuring circuit modules. Which servo CPU is to control the individual axes must also be defined. MD 200* is used to specify at which measuring circuit module or encoder (measuring circuit actual value input) the actual position values are to be received and the set speed values output.

⋮	⋮	⋮	⋮
Measuring circuit module No.	Number of the actual value input (encoder)	Servo CPU No.	Servo CPU axis No. (entered automatically by the control)

Possible values: 00 00 00 00 → Axis not available on machine (permissible only with MD 564* bit 7 = 0)

01 01 01 00 → 1st measuring circuit module, 1st encoder, 1st servo CPU

.

04 02 02 00 → 4th measuring circuit module, 2nd encoder, 2nd servo CPU

.
.
.

10 03 04 00 → 10th measuring circuit module, 3rd encoder, 4th servo CPU

Caution!

No other values are allowed.

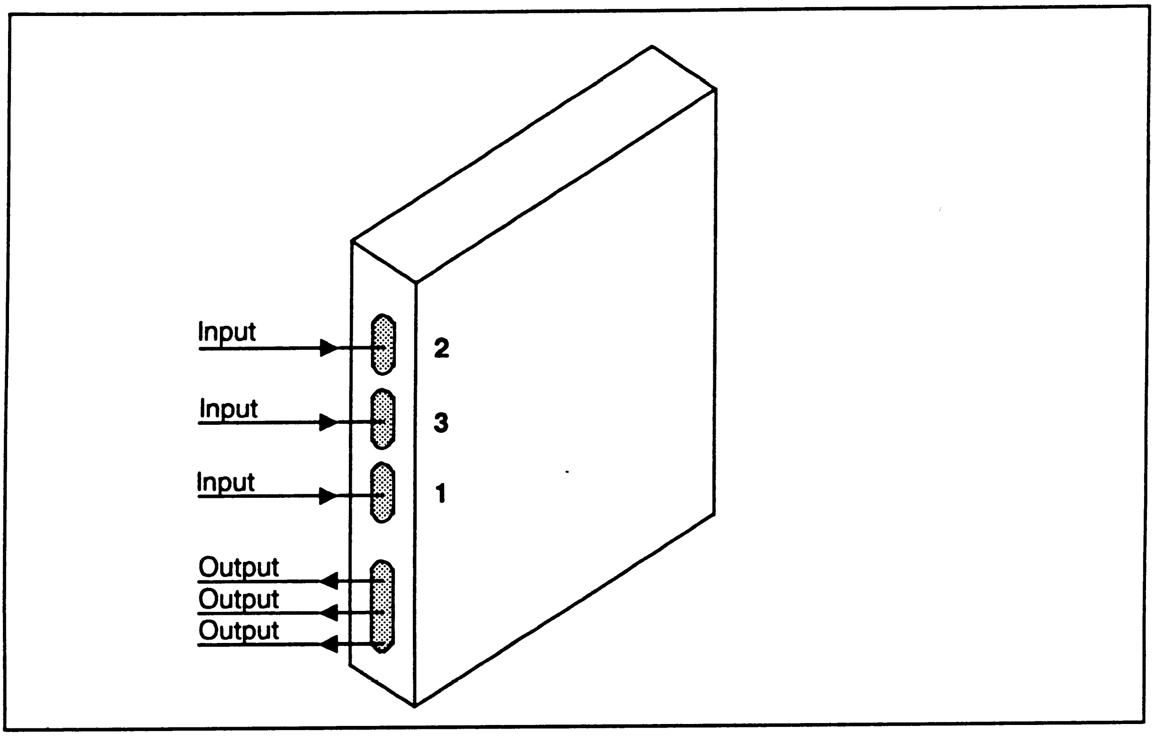
All servo CPUs and all measuring circuit modules are in the same bus so that the servo CPUs and the measuring modules each are counted through from left to right starting at 01. There can be no gaps in the numbering as the control performs the numbering automatically with Power On. This makes jumpering using jumpers or switches unneccessary.

MD No.	Control	Standard MD
2000	880 T/ M	1 0 1 0 1 0 1
2001	880 T/ M	1 0 2 0 1 0 2
2002	880 M	1 0 3 0 1 0 3

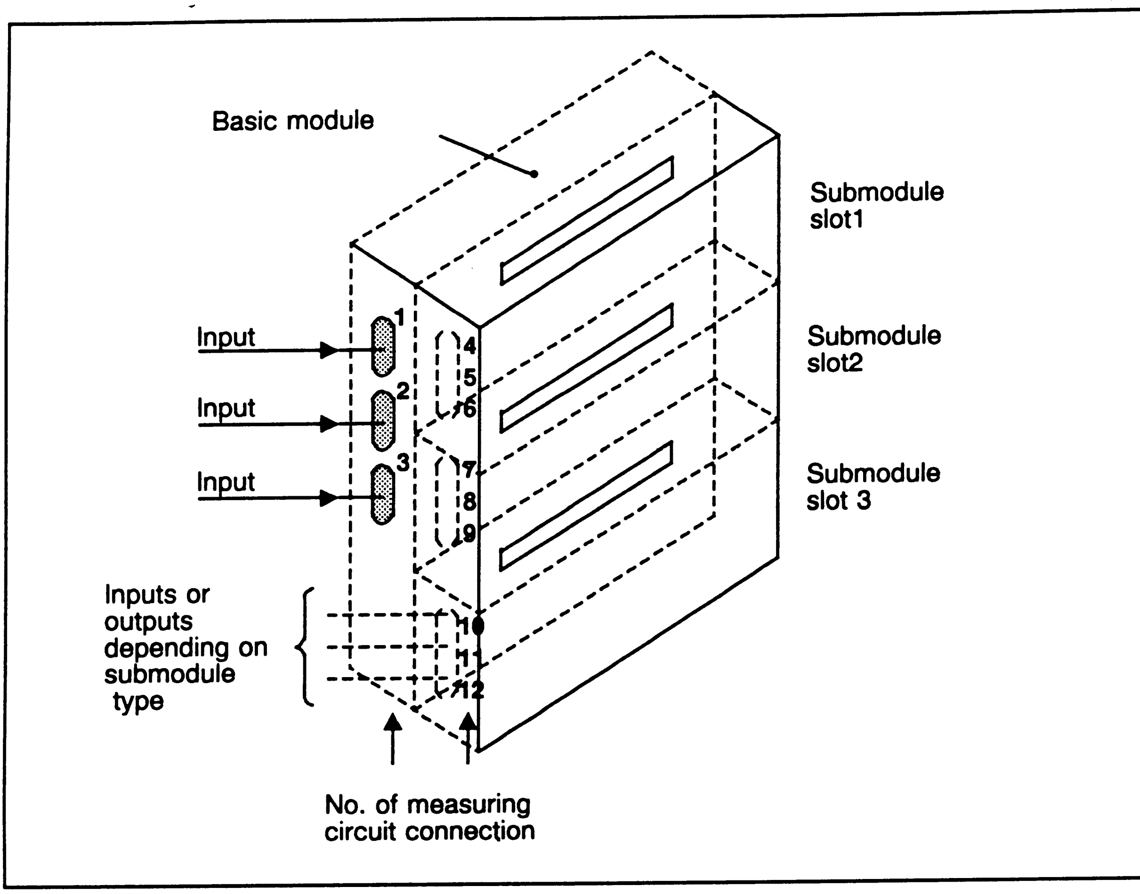
In the central controller rack measuring circuit modules do not have to be plugged in directly adjacently (free slots can be left between measuring circuit modules without gaps arising in the numbering).

Note:

- The axis actual value and the axis setpoint (analog set speed) of an axis can be connected on separate measuring circuit modules. However, the axis actual value and the axis setpoint of one axis must not be distributed to measuring circuit modules of different types (SPC measuring circuit modules 6FX1121-4B. and HMS measuring circuit modules 6FX1145-6BA..) .



Assignment of inputs/outputs on SPC measuring circuit modules



Assignment of inputs/outputs on HMS measuring circuit modules (with setpoint submodules)

204*	Exact stop limit coarse		204*
Standard value	Lower input limit	Upper input limit	Units
40	0	16 000	units (MS)

Active: When all channels of the mode group are in STOP state

A higher value may be entered in the "exact stop limit coarse" than in the "exact stop limit fine". Consequently, block change to the next machining block is initiated correspondingly earlier.

If this function is not required, it may be disabled by inputting equal exact positioning values in both machine data (coarse and fine stop).

The exact stop limit coarse is active with the following:

- G00
- Block ahead of G04
- Block ahead of setting data
- Block ahead of which only auxiliary functions are programmed
- Single block without G60/G09
- Jog
- Incremental feed
- Program end

Note:

The exact stop limit coarse is not approached in continuous-path operation G64 (exception: G00 G64 → exact stop coarse). There is no sequential error as a result of a large number of consecutive positioning operations since the closed-loop position controller is not "shut down" as a result of the exact stop limit, but instead the second block is already processed ahead of the end position of the first block.

The actual travel is now:

Remainder of first block plus second block etc. If the axis remains stationary for an instant, e.g. because another axis is about to move or because there is no axis movement in this program block, the following error is adjusted to 0 and the axis remains precisely in position.

208*	Exact stop limit fine		208*
Standard value	Lower input limit	Upper input limit	Units
10	0	16 000	units (MS)

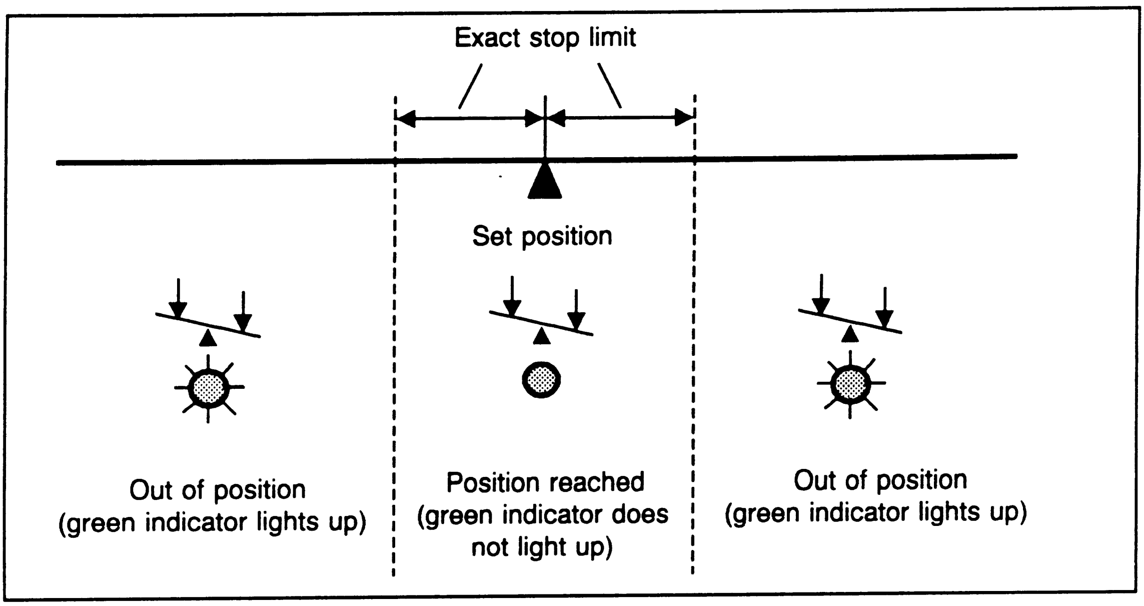
Active: When all channels of the mode group are in STOP state

A traversing movement is complete when the axis has reached the set position +/- of the entered exact stop limit fine.

If the set position is not within this range, the position control light remains lit and the block is considered incomplete.

Remedy

e.g. drift compensation



The exact stop limit fine is active with the following:

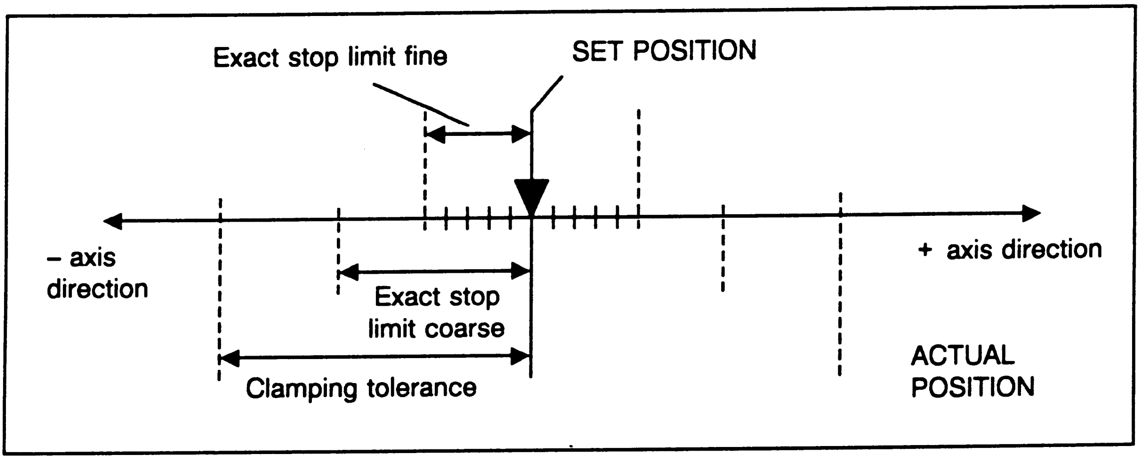
- G09/G60
- Block ahead of G33

Note

In continuous path mode (G64) neither the exact stop limit coarse nor the exact stop limit fine is approached (exception: G00).

212*	Clamping tolerance		212*
Standard value	Lower input limit	Upper input limit	Units
100	0	16 000	units (MS)

Active: When all channels of the mode group are in STOP state



The NC monitors the position at rest (holding of position). Alarm 112* is displayed if the clamping tolerance is exceeded after the delay for position monitoring (NC MD 156 or NC MD 372*).

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 is no longer held in position by the NC. The interface controller must hold the axis in position itself by means of clamping. As a result, the clamped axis may be forced out of position due to mechanical influences.

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

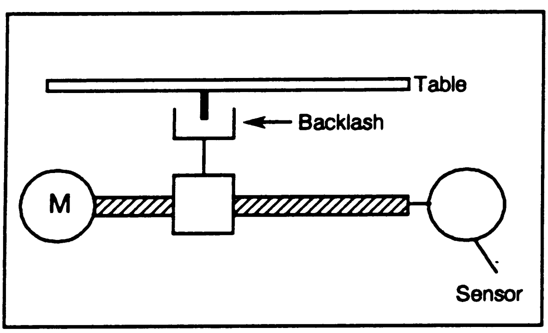
The clamping tolerance entered must be **greater than the exact stop limits fine and coarse.**

220*	Backlash compensation		220*
Standard value	Lower input limit	Upper input limit	Units
100	-16 000	16 000	units (MS)

Active: When all channels of the mode group are in STOP state

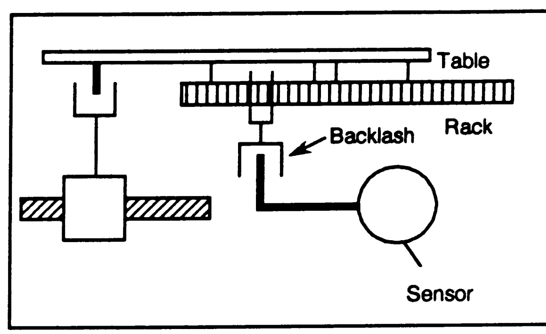
In the case of axes with indirect measuring systems, mechanical backlash results in corruption of the traverse path. On reversal of the direction of movement, traverse is either shortened or extended by the amount of backlash, depending on the design.

Positive backlash (normal conditions)



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

Negative backlash



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

With positive backlashes the compensation value (amount of backlash) is entered as a positive value and with negative back lashes it is entered as a negative value. The backlashes in NC MD 220* are compensated by the control every time the axis in question changes direction (in all modes and types of interpolation).

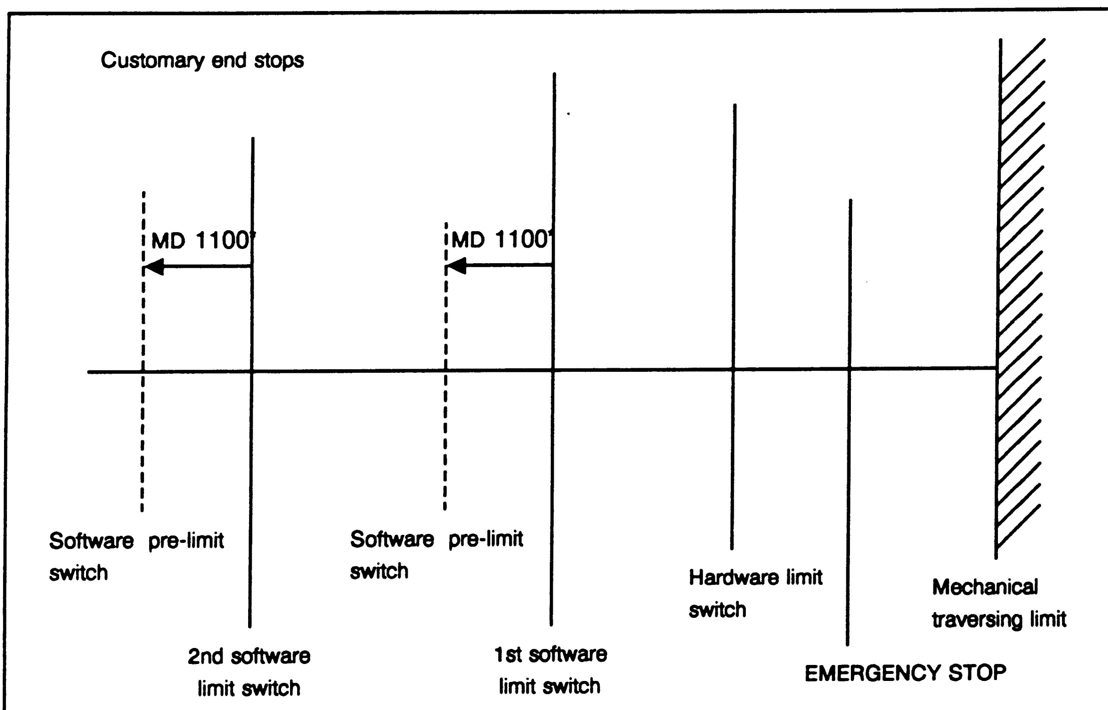
224*	First software limit switch (positive direction)		224*
Standard value	Lower input limit	Upper input limit	Units
99 999 999	-99 999 999	99 999 999	units (MS)

Active: When all channels of the mode group are in STOP state

A software limit switch may also be used instead of the customary range limit switch (hardware limit switch). The absolute position of the positive range limit for each axis is input.

The software limit switches cannot fulfil their function properly until the reference point has been approached.

The software limit switches are always approached at the rate entered under NC MD 1, unless a slower rate has been selected in the current motion block. Braking to zero speed is performed so far ahead of the software limit switch that this switch is precisely reached but not overrun (in jog mode). In certain circumstances the software limit switch can be overrun (see NC MD 1100*).



Note:

In the case of axes traversed in interpolation, all axes are shut down if the range limit of one axis has been reached. However, stopping without contour violation is only guaranteed if NC MD 5003 bit 7 ("No deceleration at limit switch") has not been set, i.e. during braking over acceleration ramp.

228*	First software limit switch (negative direction)		228*
Standard value	Lower input limit	Upper input limit	Units
-99 999 999	-99 999 999	99 999 999	units (MS)

Active: When all channels of the mode group are in STOP state

Same significance as for NC MD 224* but for traversing limit in negative direction.

232*	Second software limit switch (positive direction)		232*
Standard value	Lower input limit	Upper input limit	Units
99 999 999	-99 999 999	99 999 999	units (MS)

Active: When all channels of the mode group are in STOP state

A second limit switch position in the positive direction may be specified. Which of the two software limit switches 1 or 2 is to be active is selected by the PLC by means of an interface signal (Interface Description Part 1 DB 32 DL $_{K+1}$/bit 1).

e.g. DB 32 DL1 Bit 1 = 0 .... First software limit switch active for 1st axis
Bit 1 = 1 .... Second software limit switch active for 1st axis

Example of application:

Reduction of permissible traversing range with tailstock in position.

236*	Second software limit switch (negative direction)		236*
Standard value	Lower input limit	Upper input limit	Units
-99 999 999	-99 999 999	99 999 999	units (MS)

Active: When all channels of the mode group are in STOP state

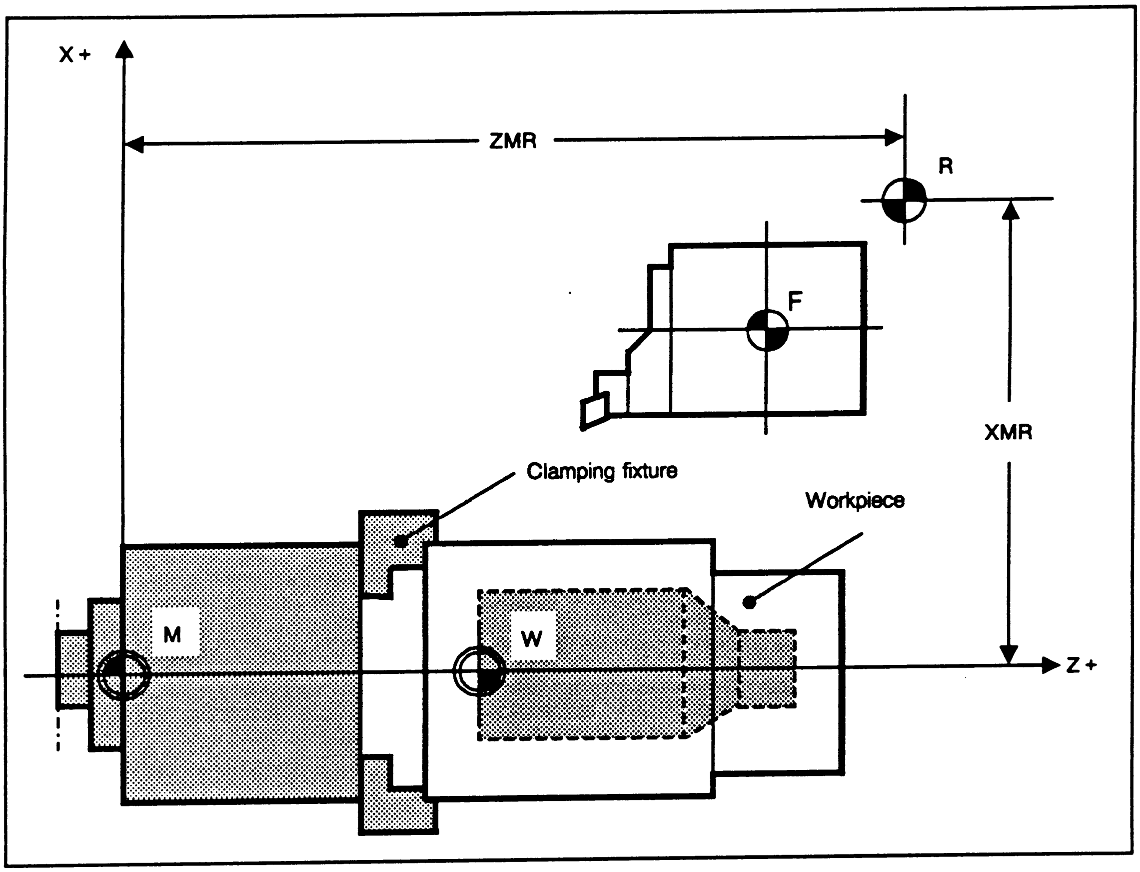
Same significance as for NC MD 232* but in negative direction. Selection by PLC program DB 32 DL$_{K+1}$/bit 0 (see Interface Description Part 1).

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

Active: When all channels of the mode group are in STOP state

The difference between the absolute machine zero and the fixed reference point is entered for the relevant axis. These values are set as actual values for approach to the reference point (see Section 11.3 for reference point approach).

Also see NC MD 1808* if an absolute encoder is used.



Example: lathe

F	...	Slide reference point
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		244*
Standard value	Lower input limit	Upper input limit	Units
0	-99 999 999	99 999 999	units (MS)

Active: When all channels of the mode group are in STOP state

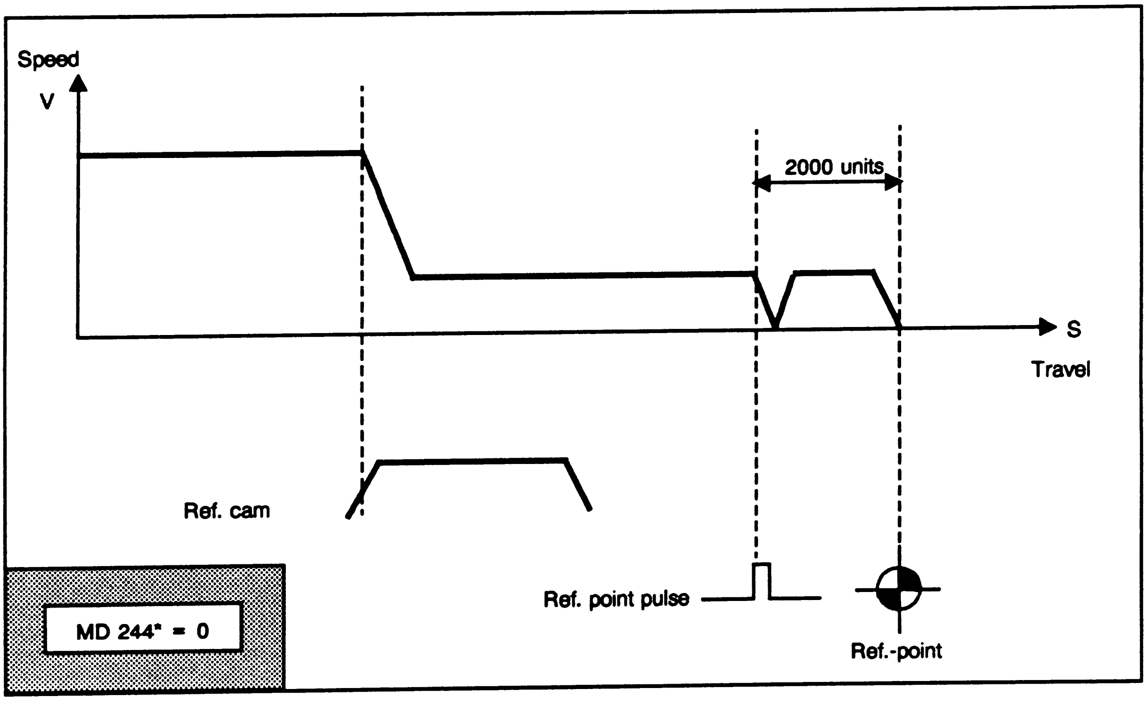
The measuring-system reference points may be shifted by means of reference point shift. Consequently, instead of mechanical shifting or rotating of the measuring system (and thus the "deceleration" cam), the reference point may be shifted electrically up to ± 9 999 999 units. The reference point shift path is traversed at the cutoff speed (NC MD 284*) which must already be reached at the operating cam (for reference point approach also see Section 11.3).

The reference point shift is also effective with automatic reference point approach (NC MD 560* bit 6).

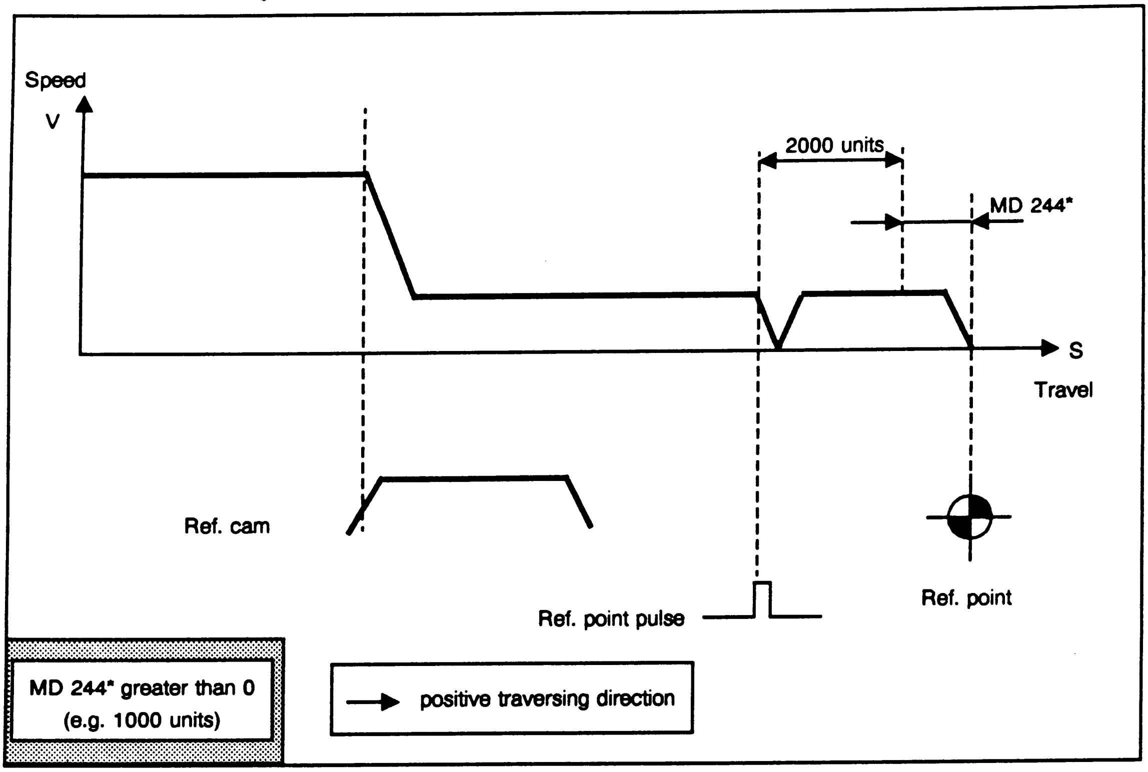
Without reference point shift, the reference point is 2000 units behind the first zero mark after the operating cam has become free again. This does not apply to C axes that are assigned to a spindle (NC MD 461* ≠ 0).

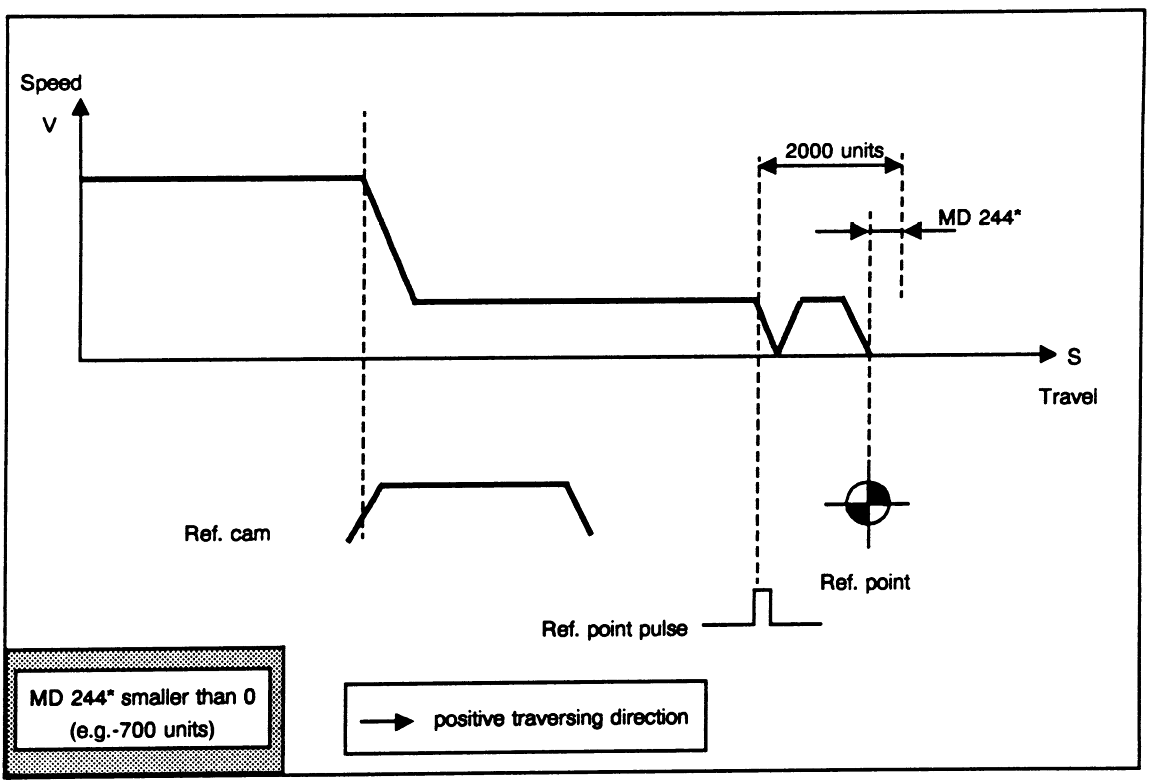
With positive input the axis travels in the positive direction by the input value beyond the normal reference point (2000 units after zero mark).

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



If a C axis assigned to a spindle, the zero offsets 244* and 459* must be identical.


Speed
V
2000 units
MD 244*
S
Travel
Ref. cam
Ref. point
Ref. point pulse
MD 244* greater than 0
(e.g. 1000 units)
positive traversing direction


Speed
V
2000 units
MD 244*
S
Travel
Ref. cam
Ref. point
Ref. point pulse
MD 244* smaller than 0
(e.g.-700 units)
positive traversing direction

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

Active: When all channels of the mode group are in STOP state

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

The absolute position of the crosshair must be input.

252*	Servo gain factor (Kv factor)		252*
Standard value	Lower input limit	Upper input limit	Units
1 666	0 or 166	10 000	0.01 s^{-1}

Active: When all channels of the mode group are in STOP state

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 as well. Strictly speaking therefore, a distinction must be made between a "desired servo gain factor" (above NC 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:

- Max. speed (MD 280*)
- Multgain (NC MD 260*)
- Maximum laod speed (NC MD 256*)
- Tacho-generator compensation at speed controller
- Tacho-generator at drive

The input value of 1 666 is equivalent to a servo gain factor of 1.

Notes:

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

Any deviations will result in contour defects!

Only axes **never** contributing to continuous-path operation may be defined with different values (determination of servo gain factor). The actual servo gain factor can be checked using the following error (in the service displays). Please note that drift compensation must be performed before the check is made.

An input of "0" opens the position control circuit.

256*	Maximum load speed		256*
Standard value	Lower input limit	Upper input limit	Units
10 000	1	9999 9999	Special format

Active: When all channels of the mode group are in STOP state

The maximum load speed is derived from the maximum motor speed, the mechanical gearbox and/or the spindle pitch.

The units depend on NC MD 1800*. If an "inch" position control resolution was selected there, the unit in MD 256* is if $\left[\frac{\text{inch}}{\text{min}}\right]$ a metric position control resolution was selected, the unit in MD 256* is $\left[\frac{\text{mm}}{\text{min}}\right]$. The unit with rotary axes is always $\left[\frac{\text{degrees}}{\text{min}}\right]$.

Examples:

a) Linear axis
Max. motor speed n_{max} = 3000 rpm; spindle pitch s = 10 mm
Gear r = 1:2 (2 revolutions of the motor = 1 revolution of the leadscrew)

$$\Rightarrow \text{Max. load velocity} = n_{max} \cdot s \cdot r = \frac{3000 \cdot 10}{2} = 15000 \left[\frac{\text{mm}}{\text{min}}\right]$$

b) Rotary axis
n_{max} = 4000 rpm; r = 1:4

$$\Rightarrow \text{Max. load speed} = n_{max} \cdot r \cdot 360 \text{ degrees} = 360\,000 \left[\frac{\text{degrees}}{\text{min}}\right]$$

Like the multgain (MD 260*), the maximum load speed serves the purpose of matching the controlled system to the Kv factor (MD 252*). Only the quotient of these two data is used in this context. In the event of accuracy problems or restrictions imposed by the input limits, the factors between MD 256* and 260* can be reduced or enlarged. See description of MD 260* for an example.

260*	Multgain		260*
Standard value	Lower input limit	Upper input limit	Units
8 000	1	9999 9999	mV

Active: When all channels of the mode group are in STOP state

The multgain factor serves to match the controlled system to the servo gain factor specified via NC MD 252*. Multgain is a strict multiplication factor for the servo gain factor entered and should be used for **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.

Note:

- Matching the actual servo gain factor using the servo gain factor NC 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.

  U_{max} is the set speed voltage at maximum motor speed.
  The following is valid: Multgain = U_{max} [mV]

  Example 1: U_{max} = 8.345 V ➡ Multgain = 8345

  In the event of accuracy problems or restrictions imposed by the input limits, the factors between NC MD 256* and 260* can be reduced or enlarged.

  Example 2: U_{max} = 6.3585 V; MD 256* = 15000

  In order to permit entry of U_{max} with full accuracy, an enlarging factor of 2 is applied:

  ⇒ MD 260* = Multgain = 12717; MD 256* = 30000.

- Also see NC MD 256* (maximum load speed).

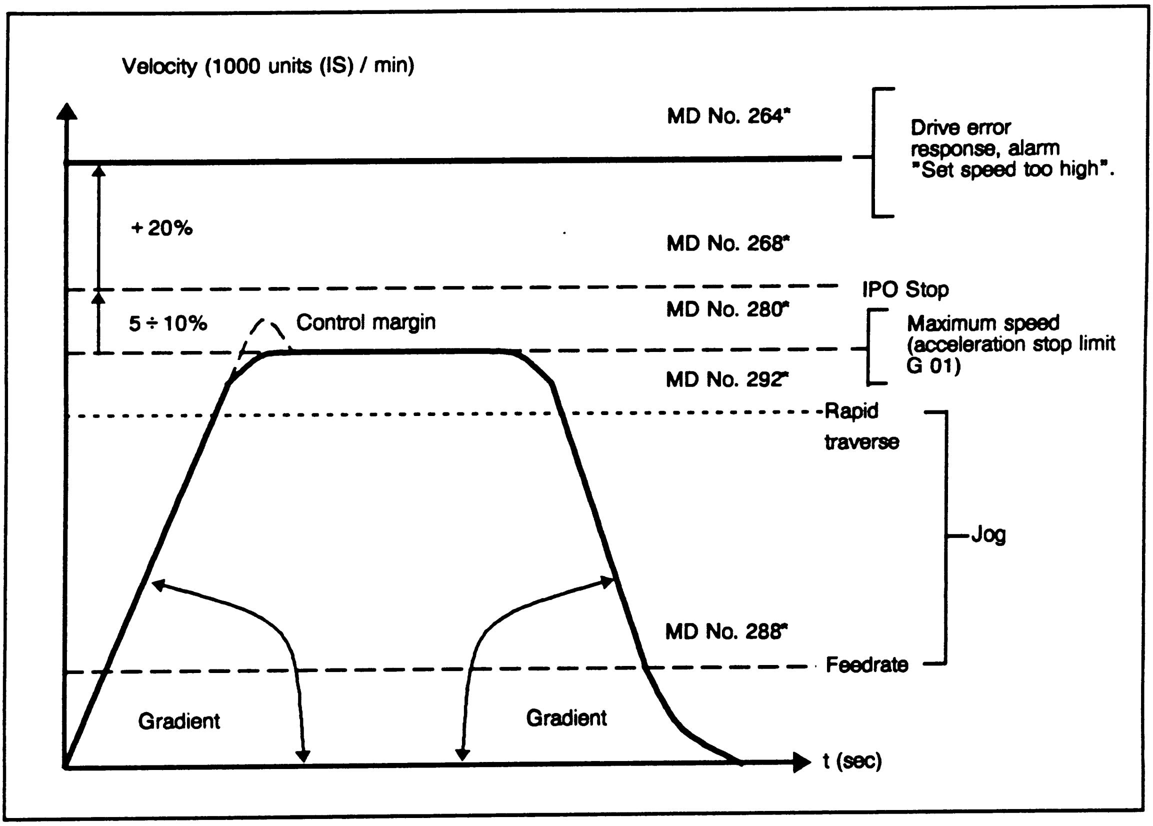
264*	Threshold for drive error		264*
Standard value	**Lower input limit**	**Upper input limit**	**Units**
9 600	0	15 000	VELO

The specified set speed is monitored. Alarm 156* is activated if the specified set speed is too high (measuring circuit and drive error).

The input value must be greater than the highest definition amount entered under NC MD 268* for the maximum set speed.

Guide value

Approx. 20 % higher than NC MD 268*

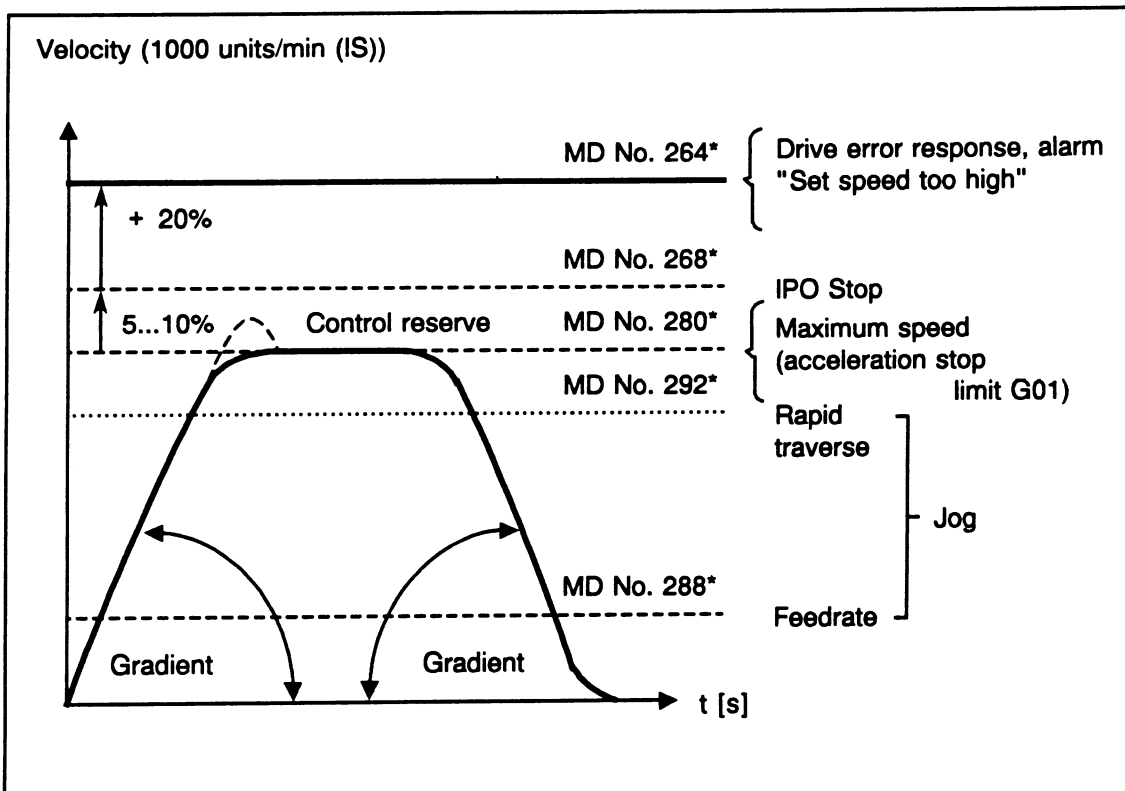


268*	Maximum set speed (IPO Stop)		268*
Standard value	Lower input limit	Upper input limit	Units
8 192	0	8 192	VELO

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 10 V).
Alarm 104* (DAC limit reached) is given when the limit is exceeded.

Caution!

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



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

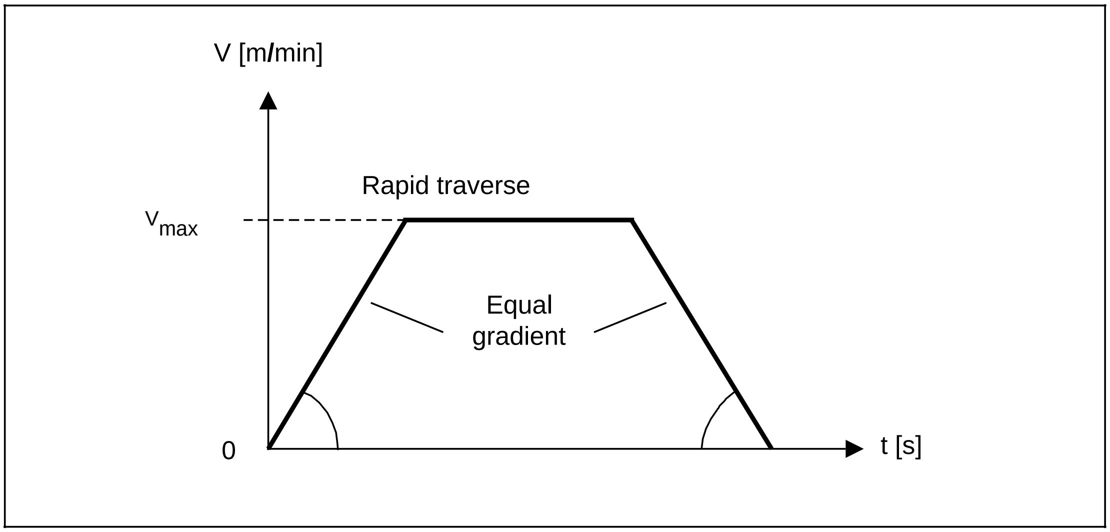
Active: When all channels of the mode group are in STOP state

The temperature drift of analog electronic components (primarily in the motor control unit) causes the axes to wander from their set position until the counter-setpoint is so great on account of the existing following error that it corresponds to the temperature drift.

If the green LED  no longer goes off although the axis is turning, the following error caused by the drift has taken on such a large value that a drift compensation has become necessary otherwise the axes will no longer be able to be moved. Drift compensation must be entered manually and modified until the following error at rest has settled at approximately 0.

276*	Acceleration		276*
Standard value	Lower input limit	Upper input limit	Units
50	**1**	**16 000**	**$\frac{10\ mm}{s^2}$**

Active: Immediately



Acceleration and braking characteristic

The axes need not be set to equal acceleration values. The controller assumes the lowest acceleration value of the interpolating axes involved. The values also apply to deceleration (braking).

The acceleration in NC MD 276* is active on every acceleration or braking process (speed change) of the axis.

Exception:

On reference point approach the axis is braked as fast as possible on reaching the zero mark. For this reason a small value must be selected for the reference point creep speed (NC MD 284*). If alarms occur the axis concerned is also braked as fast as possible (also see alarm description).

Note:

Values of 50 ... 150 (= 0.5 ... 1.5 m/s^2) are usual for a standard machine.

For rotary axes the angular acceleration must be entered here.

280*	Maximum speed		280*
Standard value	Lower input limit	Upper input limit	Units
10 000	0	10 720 000	1000 units / min (IS)

Active: When all channels of the mode group are in STOP state

The limit speed up to which the axis may accelerate (**rapid traverse limit**) is entered. Traversing is performed at this speed with programmed rapid traverse G00. All programmed and entered speeds are referenced to this MD.

The maximum permissible speed depends on the position control resolution (also see Section 11.7.7).

284*	Reference point cutoff speed		284*
Standard value	Lower input limit	Upper input limit	Units
300	0	10 720 000	1000 units / min (IS)

Active: When all channels of the mode group are in STOP state

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 (see NC MD 244*). The feedrate override switch is not taken into account, except in the first position (0 %). The value input must not exceed the maximum speed (NC MD 280*) (for reference point approach also see Section 11.3).

Guide value:

A reasonable upper limit is 1 m/min, but values between 100 and 500 mm/min are better, depending on the servo gain factor.

In the case of reference point approach with "distance-coded reference point markers", the reference point cutoff speed must not exceed a maximum value (also see Section 11.16).

288*	Feed in jog mode		288*
Standard value	Lower input limit	Upper input limit	Units
2 000	**0**	**10 720 000**	**1000 units / min (IS)**

Active: When all channels of the mode group are in STOP state

The input value applies to travel in JOG mode with the feedrate override switch in the 100% position. The value entered must not exceed the max. speed (NC MD 280*).

292*	Rapid traverse in jog mode		292*
Standard value	Lower input limit	Upper input limit	Units
5 000	**0**	**10 720 000**	**1000 units / min (IS)**

Active: When all channels of the mode group are in STOP state

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.

The value entered must not exceed the max. speed (NC MD 280*).

This value is not used for programmed rapid traverse G00. Programmed rapid traverse G00 is specified by the maximum speed NC MD 280*.

Guide value:

A value lower than rapid traverse G00 should be selected to make allowance for the operator's response time.

296*	Reference point approach speed		296*
Standard value	Lower input limit	Upper input limit	Units
10 000	**0**	**10 720 000**	**1000 units / min (IS)**

Active: When all channels of the mode group are in STOP state

If the direction key leading to the reference point (selectable using NC MD 564*) is pressed in "reference point approach" mode, the axis accelerates to the reference point approach speed (exception: axis already at deceleration cam or automatic reference point approach selected, also see NC MD 244*).

The value entered must not exceed the max. speed (NC MD 280*).

For reference point approach also see Section 11.3.

300*	Incremental feedrate		300*
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**10 720 000**	**1000 units / min (IS)**

Active: When all channels of the mode group are in STOP state

The input feedrate is active only with incremental feed (INC1 ... 10 000).

The value entered must not exceed the max. speed (NC MD 280*).

The feedrate during traversing with the handwheel in modes INC1 ... 10 000 and with the DRF function is determined exclusively by the handwheel.

With ELG: "On-the-fly synchronization between LA/FA".

The following axis traverses the established path diference to the synchronization position as an overlay movement at incremental velocity.

304*	Interpolation parameter		304*
Standard value	Lower input limit	Upper input limit	Units
1	**0**	**3**	**-**

Active: In next block

In the case of circular movements (G2/G3) and thread cutting (G33, G34, G35), the individual axes must be assigned an interpolation parameter:

0	=	No interpolation parameter	
1	=	Interpolation parameter I	for 880 M/T
2	=	Interpolation parameter J	for 880 M
3	=	Interpolation parameter K	for 880 M/T

Standard MD: X axis I
Y axis .... J
Z axis .... K

A number of axes may bear the same interpolator name.

Programming for same interpolator names:

G2 X5 C10 J20 J20 LF

Assignment: 1st interpolator to 1st programmed axis
2nd interpolator to 2nd programmed axis

Axes without interpolator names cannot perform circular movements, helical movements and thread cutting.

308*	Cutoff frequency of C axis encoder		308*
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**16 000**	**kHz**

Active: After POWER ON

The cutoff frequency of the actual-value encoder for the C axis is entered in the machine data, the value being taken from the manufacturer’s documentation.

If the cutoff frequency is exceeded it is possible for pulses from the encoder to be lost, which will cause an error in the actual value acquisition. The C axis mode cannot be used any more.

If the critical frequency is not exceeded, the C axis is automatically resynchronized. The interface signal "spindle synchronized" changes from 0 1.

- The defined servo gain factor (MD 252*) must correspond to the actual condition if the cutoff frequency monitoring is to function correctly (check following error).
- If the same encoder is used for the spindle and C axis, only the spindle encoder frequency (NC MD 462*) is monitored.

312*	P-component feedforward control factor		312*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 000**	**0.1 %**

With the help of the feedforward control, the synchronous error resulting from the following error between the following axis and the leading axes should be greatly reduced.

In the feedforward control, the part set value multiplied by the feedforward control factor is taken directly to the speed controller input. It is applied to the input of the position controller directly in the case of static feedforward control or with a PT-1 element delayed by the time constant specified in machine data (392*) in the case of dynamic feedforward control. If the feedforward control factor 0 is entered, no feedforward control is calculated for the relevant axis.

The feedforward control factor can be adapted according to the machine stability and the acceleration/deceleration of the axis. The following error reduces in accordance with the feedforward control factor when feedforward control is applied.

A factor of 1000 in the steady state corresponds to a following error of virtually 0. Note however that overshoots occur with this setting. If this is not desired, a correspondingly lower value must be entered and the time constant for the feedforward control must be entered in MD 392*.

The effect of this MD can be seen from the service display by considering the following error.

There are two types of feedforward control:

Static feedforward control is set with the following parameter assignment:

- Feedforward control factor 1 MD 312* 1000
- Time constant = 0 MD 392* = 0.

With the aid of static feedforward control, the following error can be fully compensated with constant speeds (i.e. with steady-state operation). However, such a setting causes a marked positional overshoot during the acceleration phase.

Overshooting during positioning operations is undesirable, however, with many applications. The following setting (**dynamic feedforward control**) is recommended in such instances:

Feedforward control factor: $K_{VOR} = 0.66$

Time constant for feedforward control: $T_{VOR} = 0.3$ to $0.5 \cdot T_n$

T_n = Rise time constant of speed control circuit

Such a setting reduces the following error by about one-half.

When using position-controlled leading axes and feedforward control, the feedforward control parameters (feedforward control factor and time constant) of all axes involved on the contour must be set the same, as is the case with the servo gain (K_V) factors.

Note the following points when using dynamic feedforward control:

1. Rigid machine behaviour
2. Dynamic drives and axes (AC drives)
3. Short position control cycle (2.5 ms)
4. Set servo gain (K_V) factor as high as possible (in order to improve disturbance characteristic)

Note:

The "Feedforward control" Option must be available.

316*	Pointer compensation (+)		316*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**249**	**MD offset**

Active: After warm restart

The NC activates leadscrew error compensation after reaching the reference point. Consequently, the CNC must be informed by means of MD 316* as to which of the 1000 - 2000 possible compensation points represents the reference point for the axis in question (see Section 11). If leadscrew error compensation is not direction-dependent, the same value 316* must also be entered in MD 320* (for leadscrew error compensation see Section 11).

320*	Pointer compensation (-)		320*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**249**	**MD offset**

Active: After warm restart

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 (for leadscrew error compensation see Section 11).

324*	Distance between two leadscrew error compensation points		324*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**99 999 999**	**units (MS)**

Active: After POWER ON

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

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

(for leadscrew error compensation also see Section 11).

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

Active: After POWER ON

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 (for leadscrew error compensation also see Section 11).

332*	Contour monitoring tolerance band		332*
Standard value	Lower input limit	Upper input limit	Units
1 000	**0**	**99 999 999**	**units (MS)**

Active: When all channels of the mode group are in STOP state

The operating principle of contour monitoring is based on a continuous comparison of the measured following error of the axes and that calculated in advance from the NC partial setpoint.

The model used to calculate the following error in advance emulates the dynamic response of the position control including feedforward control.

If the difference between the measured and pre-calculated following error exceeds the set tolerance band for longer than the settable time (MD 1200*), the control of the axis concerned is shut down and the "contour monitor" alarm message is issued.

In order to prevent erroneous tripping when stationary, a minimum speed above which the contour monitor is to be active can be stated in MD 336*.

The contour monitor can be deselected with NC MD 1820* bit 7 for installation purposes.

336*	Contour threshold speed		336*
Standard value	Lower input limit	Upper input limit	Units
5	0	16 000	1000 units / min (IS)

Active: When all channels of the mode group are in STOP state

The speed above which contour monitoring is to be active is input. No contour monitoring is active below this axis-specific threshold speed. With the axis at a standstill, the zero-speed control monitors excessive axis movements (Alarm 112*). See Section 3.8 for a detailed description.

340*	Tool change position		340*
Standard value	Lower input limit	Upper input limit	Units
0	-99 999 999	99 999 999	units (MS)

Active: After POWER ON

The following applies only to SINUMERIK 880 T:

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 NC MD makes it possible to specify a maximum retract position, e.g. to protect machine parts located behind.

344*	Rotary axis modulo value for leadscrew error compensation		344*
Standard value	Lower input limit	Upper input limit	Units
360 000	0	92 160 000	units (MS)

Active: When all channels of the mode group are in STOP state

The modulo value is also active with the "C axis on-the-fly synchronization" function.
For this, the value must be set correctly according to the position control resolution
(3 600, 36 000, 360 000, 3 600 000, 36 000 000).

The value of the compensation area for a rotary axis is input here. A value of 360 000 ≙ (one revolution of the rotary table) is usually a practical value.
If values greater than 360 000 are input it is important to make sure that a corresponding number of leadscrew error compensation points is made available (for leadscrew error compensation see Section 11).

If ELG is used:

If the following axis is a rotary axis the modulo value of the rotary axis must be entered in MD 344*. The value is normally 360 000.

348*	**Minimum working range for simulation**		**348***
Standard value	Lower input limit	Upper input limit	Units
- 5 000	**- 32 000**	**0**	**mm or inch**

352*	**Maximum working range for simulation**		**352***
Standard value	Lower input limit	Upper input limit	Units
5 000	**0**	**32 000**	**mm or inch**

Active: After simulation start

The machine data specify the value range for each NC axis in simulation:
For every NC axis there is a minimum and a maximum value. Only the machine data of the axes which are being simulated are relevant. In double-slide simulation the working range is defined by the axis of the first simulation channel. The unit depends on NC MD 5002 bit 4.

It is important to make sure that the resolution is inversely proportional to the working range. If you wish to observe details of the simulation, select the working range to be as small as possible. Moreover you must select the working range so that all absolute values move within this area. The absolute value can be composed of the following components.

Absolute value = programmed value
+ settable zero offset
+ programmable zero offset
+ length correction (LK)
+ tool nose/cutter radius compensation (TNRC/CRC)

Internally the simulation space is selected to be twice as large as the working range so that circles whose centre are outside the working range can be calculated. The centre of the working range (max. working range + min. working range) / 2) of the facing axis on turning machines must be 0, i.e.: max. working range = - (min. working range).
The resolution is determined by the simulation axis with the largest working range. This gives the smallest unit of simulation with:

Example:

$$\frac{\text{max. working range} - \text{min. working range}}{32768}$$

$$\frac{+16\text{ m} - (-16\text{ m})}{32768}$$

If the display area on the screen is 550 pixels, a window size results for which the resolution (smallest unit of simulation) is exactly 1/2 pixel. This is the limit value from which the calculation precision can affect the display.

1 Pixel = 2 · (smallest simulation unit)

Window = 550 pixels = 1100 (smallest simulation unit)

$$= 1100 \cdot \frac{\text{max. working range} - \text{min. working range}}{32768}$$

$$= \frac{\text{max. working range} - \text{min. working range}}{30}$$

The smallest simulation window with which inaccuracies are not yet visible is the 30th part of the size of the working range. The smallest unit which can be displayed (= 1 pixel) then corresponds to the 16500th part of the size of the working range.

356*	Max. IKA compensation value monitor		356*
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**99 999 999**	**units (MS)**

Active: Immediately

Unanticipated machine movements that are not entirely restricted by the monitors can occur with large compensation values. For this reason the current IKA compensation value is checked according to the limiting value entered in MD 356*. If this value is exceeded, an axis-specific interface signal of the PLC is set (DB32 DRn bit 6). No other reaction occurs.

In response to the signal the PLC can then activate the follow-up mode, for instance, in order to bring the axis to a standstill.

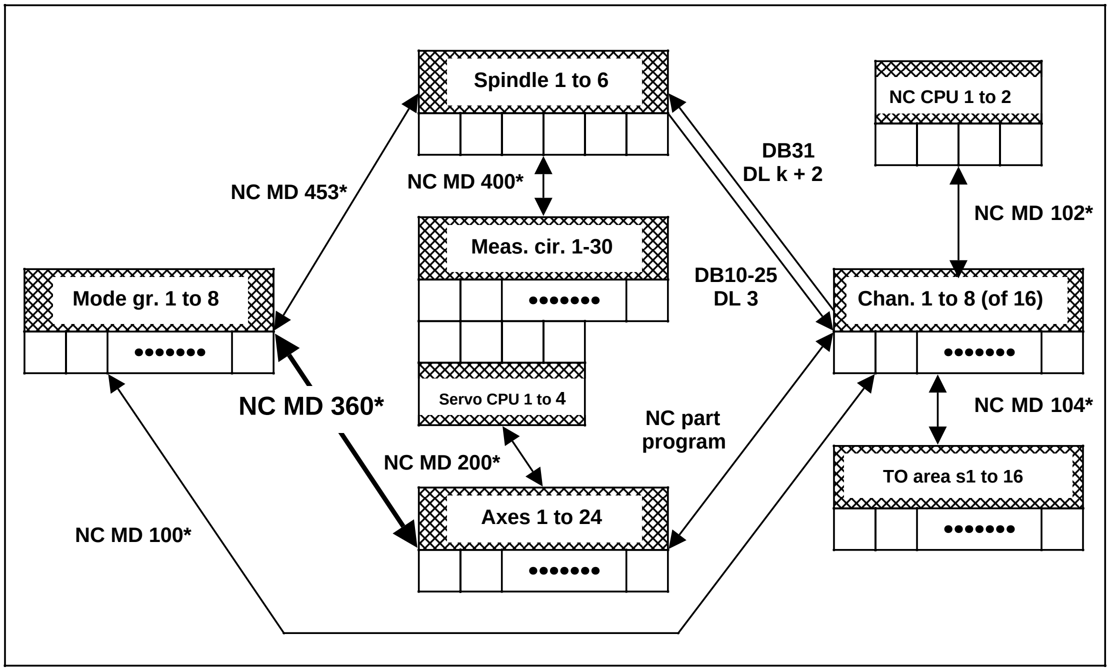
360*	Axis valid in mode group		360*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**8**	**-**

Active: After warm restart

This NC MD is used to specify axis allocation to the individual mode groups, i.e. a channel only traverses the axes of its own mode group and not those of other groups. Within any one mode group, several channels can process an axis consecutively, provided that axial synchronization of the block end value is performed for this axis (NC STOP NC START or G200 programmed) if previously traversed by another channel.

Note:

Mode groups without axes are permissible.



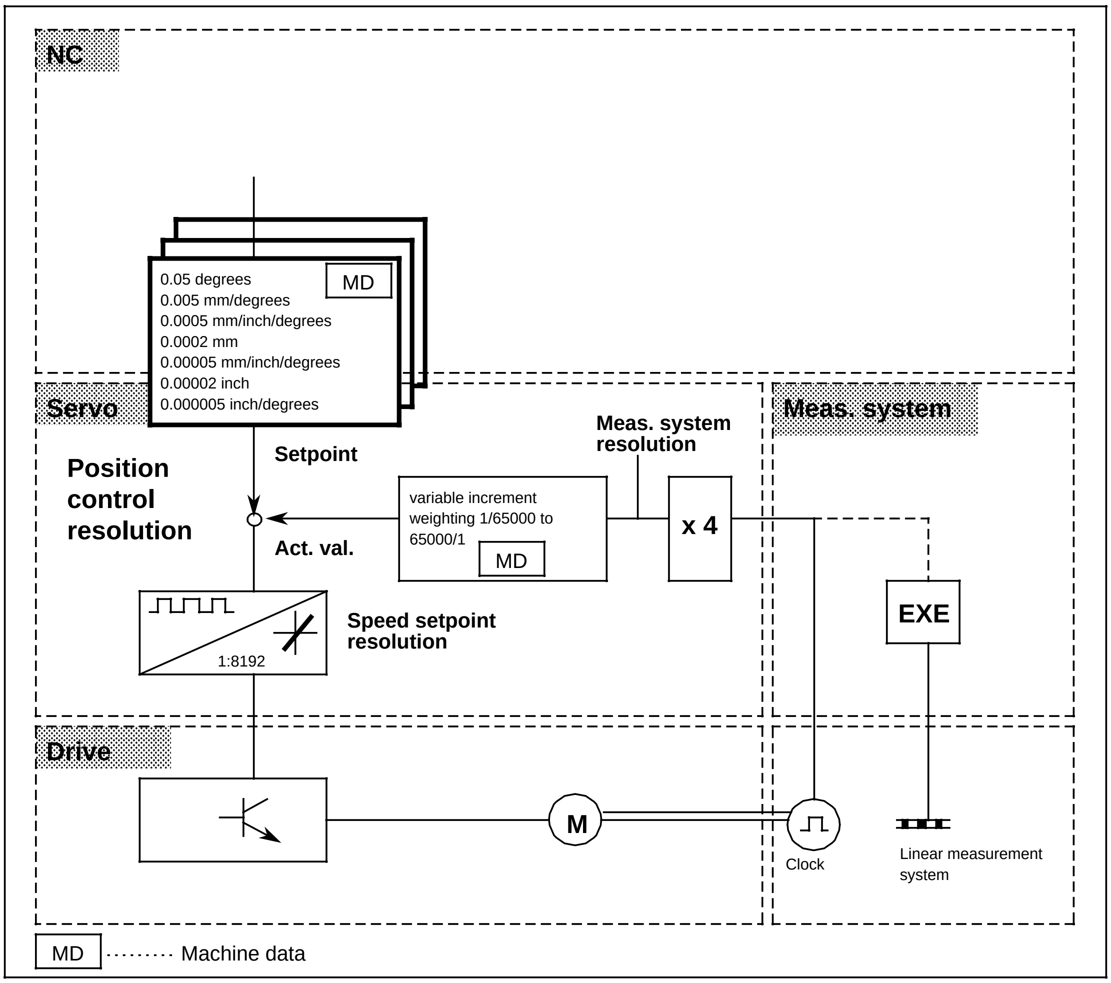
With the warm restart function, axes can be placed in another mode group without having to carry out a hardware reset. The reference point of the axes is not lost (for warm restart also see Section 11.6).

364*	Pulses for variable increment weighting		364*
Standard value	Lower input limit	Upper input limit	Units
1	**0**	**65 000**	**-**

Active: After POWER ON

For description see also NC MD 368*.

To produce a closed position control loop it is necessary to atune the pulses coming from the digital measuring system and the position control resolution to one another.



For the connection between display resolution, input resolution and position control resolution see NC MD 5002 or 1800*.

368*	**Travel for variable increment weighting**		**368***
Standard value	Lower input limit	Upper input limit	Units
1	**0**	**65 000**	**-**

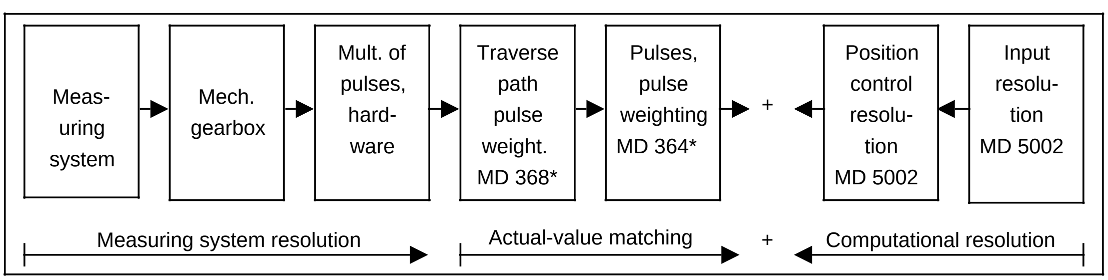
Active: After POWER ON

Variable pulse weighting (MD 364*, MD 368*):

In order to generate a closed position control loop, the pulses from the digital measuring system and the position control resolution of the controller must be matched to one another. This can be done using the following parameters:

Parameter	Symbol	Significance
Position control resolution	b	Position control resolution of control
Leadscrew pitch (lead)	s	Pitch of leadscrew or, for rotary axes, fixed at 360 degrees
Pulses per revolution	p	Number of pulses of ROD encoder per revolution
Mechanical gearbox	r	Mechanical gearbox between motor and ROD encoder (if fitted)
Grating constant	g	Period spacing on a linear scale
EXE multiplier	f	5-fold EXE means that pulses from the scale are multiplied by 5

Block diagram of the position control parameters



In order to determine machine data 364* and 368*, the pulse number for the encoder and appropriate traverse path at the machine must be known. The value of the traverse path is entered in MD 368* as a function of the positon control resolution (see examples). The pulse number of the encoder for this traverse path multiplied by all subsequent multiplications (EXE, measuring-system gearbox, hardware 4-fold multiplication on measuring-circuit module) is 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.

Examples illustrating determination of possible position control parameters:

1. Determination of MD 364* and 368*

1.1 The ROD encoder is mounted directly on the leadscrew:

Example: s = 10 mm
p = 2500 pulses per revolution
b = 1/2 10^{-3} mm

MD 368* = $\frac{s}{b} = \frac{10\ \text{mm}}{1/2\ 10^{-3}\ \text{mm}} = 20000$ MD 364* =p $*$ 4=2500 $*$ 4
=10 000

1.2 The ROD encoder is mounted on the motor with a gearbox between the motor and leadscrew:

Example 1: s = 0.2 inch
p = 1000 pulses per revolution
r = 1 : 2 (2 revolutions of motor = 1 revolution of leadscrew)
b = 1/2 10^{-4} inch

MD 368* = $\frac{s * r}{b} = \frac{0.2\ \text{inch} * 1/2}{1/2\ 10^{-4}\ \text{inch}} = 2000$ MD 364* =p $*$ 4=1000 $*$ 4
=4000

Example 2: Same values as above but b = 1/2 10^{-3} mm

MD 368* = $\frac{s * r}{b} = \frac{0.2\ \text{inch} * 25.4\ \text{mm/inch} * 1/2}{1/2\ 10^{-3}\ \text{mm}} = 5080$ MD 364*=p $*$ 4
=1000 $*$ 4=4000

1.3 A linear scale with EXE is used:

Example: g = 0.02 mm
f = 10
b = 1/2 10^{-3} mm

MD 368* = $\frac{g}{b} = \frac{0.02\ \text{mm}}{1/2\ 10^{-3}\ \text{mm}} = 40$ MD 364* =f $*$ 4=10 $*$ 4=40

1.4 A rotary axis is used:

Example: p = 18 000 pulses per revolution
f = 5
b = 1/2 10^{-3} degrees

MD 368* = $\frac{360}{b} = \frac{360\ \text{degrees}}{1/2\ 10^{-3}\ \text{degrees}} = 720\ 000$ MD 364* = p $*$ f $*$ 4=360 000

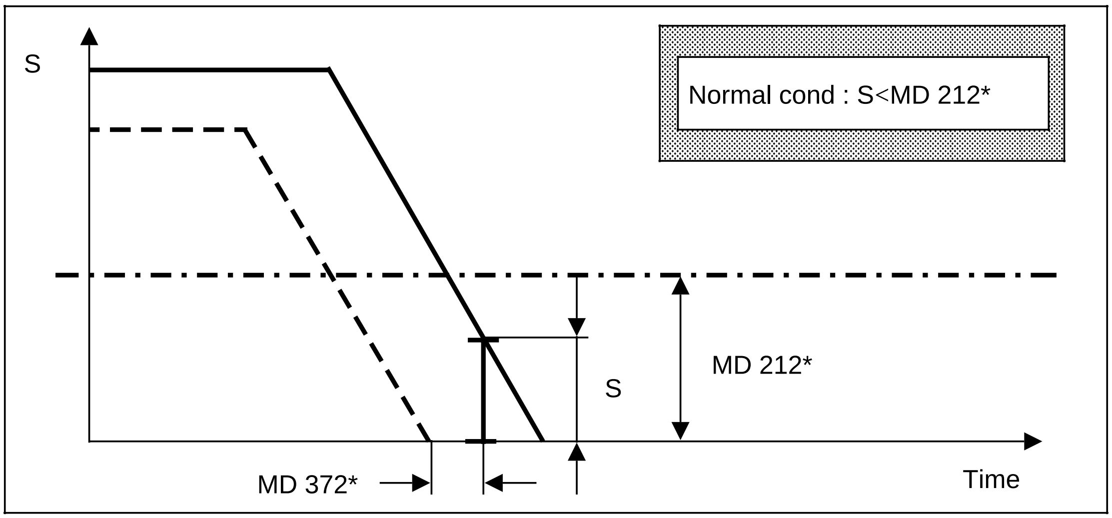
In view of the fact that the values exceed 65 000, both values must be divided by a common factor (e.g. factor=100) MD 368*=**7200** MD 364*=**3600**

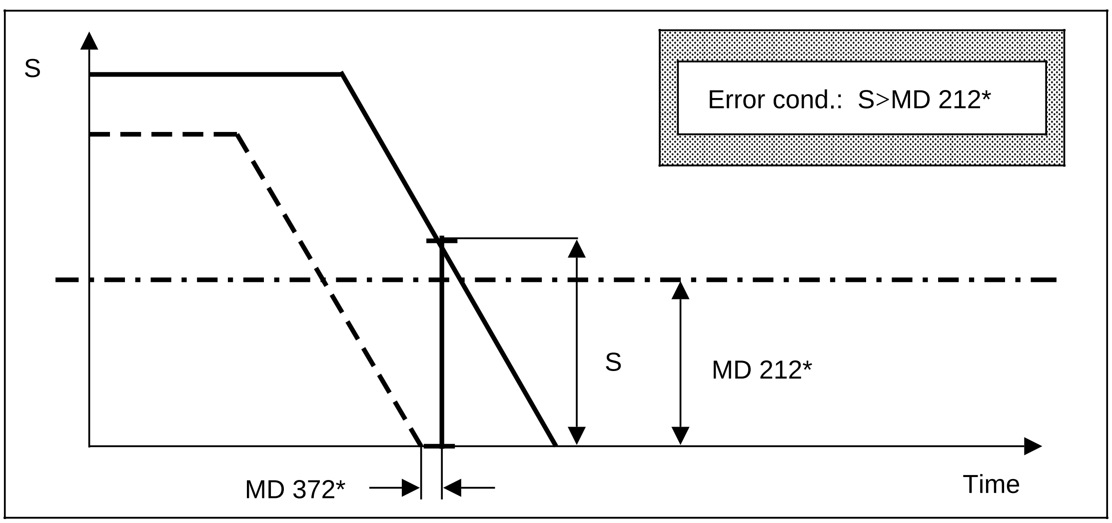
372*	Zero-speed control delay		372*
Standard value	Lower input limit	Upper input limit	Units
200	0	1 000	ms

Active: When all channels of the mode group are STOP state

The time after which clamping tolerance NC MD 212* is activated during positioning (digital zero) is entered in this datum. The time selected must be such that the maximum following error can be suppressed. If this is not the case: Alarm 112*.

If NC MD 372* = 0, the value from NC MD 156 "Servo enable cutoff delay" is assumed as the delay for zero-speed control.



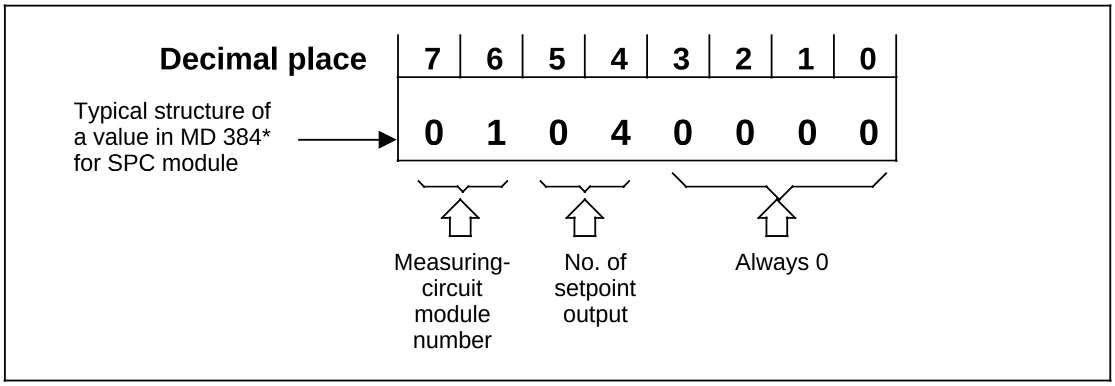


S....following error

384*	Assignment setpoint output		384*
Standard value	Lower input limit	Upper input limit	Units
see: possible values	**0**	**10 060 000**	**-**

Active: After POWER ON

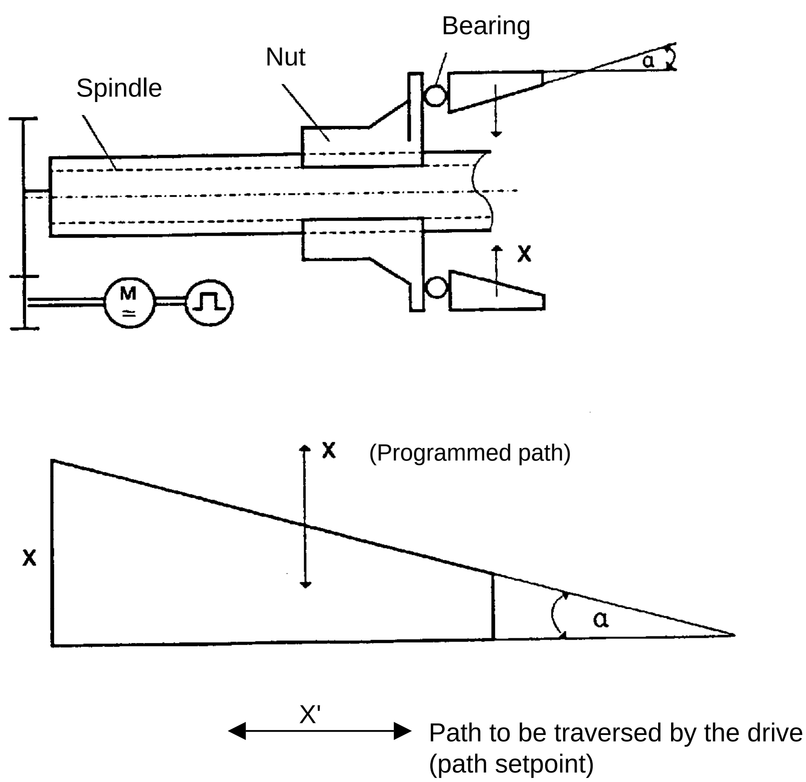
Meaning and input values of measuring-circuit module number and number of measuring-circuit connection same as NC MD 200*, but in this case valid for the assignment of the axis-specific analog set speed to a measuring-circuit module.



388*	Weighting factor		388*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**99 999 999**	**-**

Active: After POWER ON

For the manufacture of dynamically balanced parts by deformation, hammer and kneading machines are used. With these machines it can occur that a path X is programmed but a path X' must be traversed.

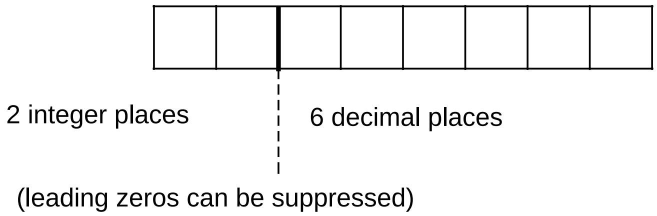


$$X' = \frac{X}{\tan}$$

X.....programmed value
X'....prepared value

The tan (not the angle) is entered as the weighting factor in NC MD 388*, for which the value range +0.00001 to +99.999999 is permissible.

The weighting in NC MD 388* is as follows:



The value zero is interpreted as weighting factor 1, and the programmed path will be the same as the path to be traversed.

Conditions:

a) All further NC MD (in units [IS]) must be input in the programmed system (X).

b) All further NC MD (in units [MS]) must be input in the prepared system (X').

c) The actual value display (not in the service displays), zero offsets, the tool offset, the PRESET offset, etc., refer to the programmed system.

d) The programmed F value refers to the programmed system.

e) If the programmed or predefined speed exceeds the permissible axis speed in the prepared system or the internal computing format, alarm 2031 "eval. factor too high/low" is set and the processing and NC start inhibited.

f) The weighing factor is active in **all** modes.

392*	Time constant for dynamic feedforward control		392*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1 000**	**0.1 ms**

In order to avoid overshoots on the axes when feedforward control is applied, the partial setpoint is applied with delay to the speed controller. This delay is set with this MD.

Input value:

Axis-specific position control cycle:	Static feedforward control (e.g. for AC drives with rise time < servo sampling time);
> Axis-specific position control cycle:	Dynamic feedforward control (for axes with rise time > servo sampling time).

See NC MD 312* for a more detailed explanation.

396*	Correction absolute encoder		396*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**±9999 9999**	**units (IS)**

Active: After POWER ON or warm restart

The offset value is determined automatically with reference point approach or can be entered by hand when synchronizing without reference point approach.

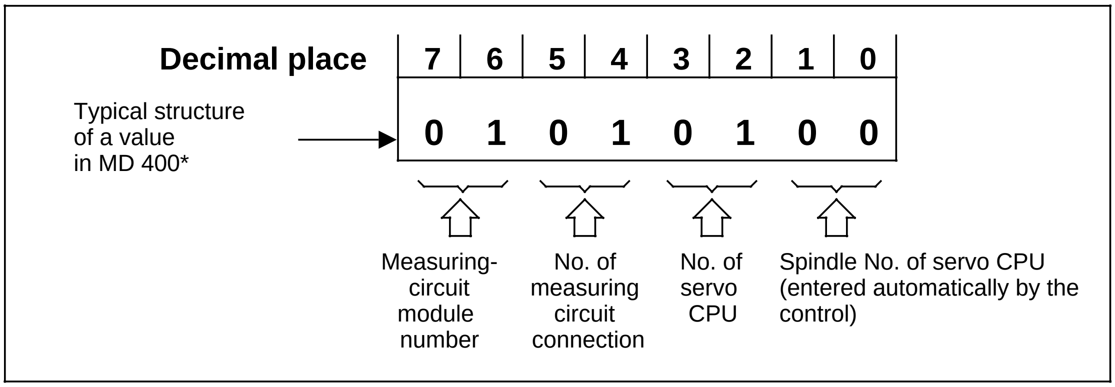
In addition, the NC machine data bits 1808* bits 0 to 3 have to be taken into account.
In the case of reference point approach with distance-coded reference point markers, the offset between machine zero and the start of the linear scale is entered (see Section 11.16).

The axis-specific values from NC MD 1100* are sorted according to their numbering.

400*	Spindle assignment, actual-value input		400*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**10 030 400**	**-**

Active: After POWER ON

Assignment of the machine's spindles to the servo CPUs and measuring-circuit modules in the system's configuration is very flexible. It is effected separately according to actual-value inputs (MD 400*) and setpoint outputs (MD 460*).



*Structure of MD 400**

Meanings of terms

- The **measuring-circuit module number** is the number of the module on the servo local bus. The modules are numbered in ascending order from left to right. The module at the far left is No. 1. If there is no spindle encoder, "00" must be entered here.

  Permitted values: 00 to 10

- The **number of the measuring-circuit connection** refers to the number of the input on the selected HMS or SPC module. The numbering of the inputs and outputs is shown in the following figure. If there is no spindle encoder, "00" must be entered here.

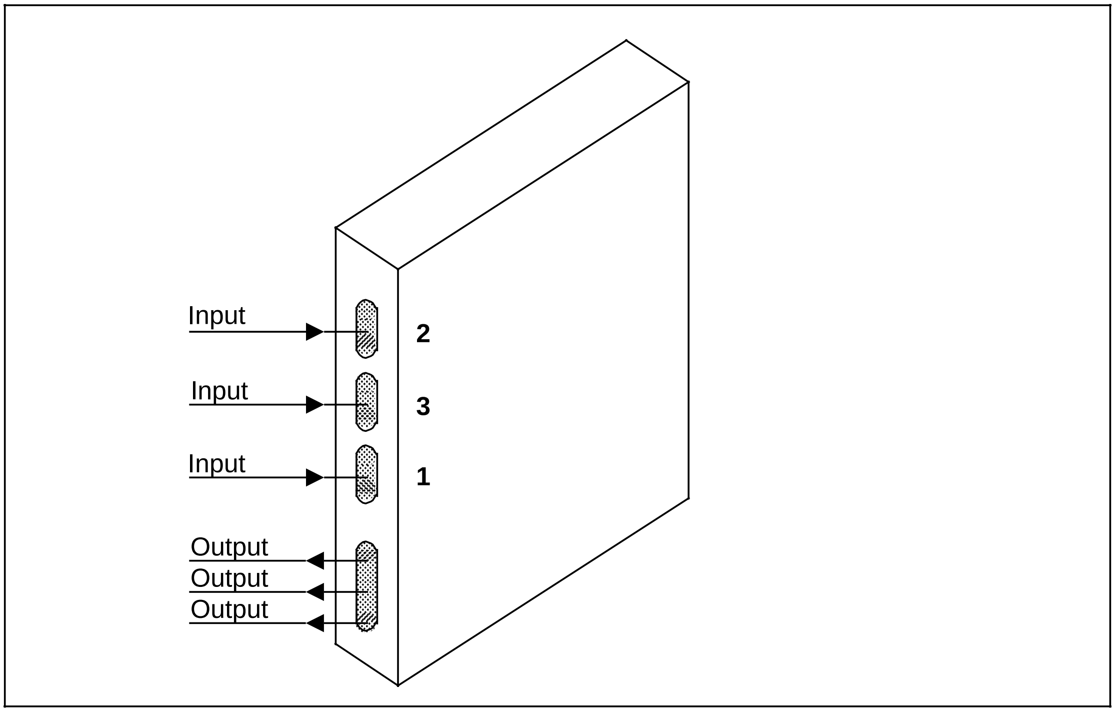
  Permitted values: 00 to 03

- The **number of the servo CPU** identifies the servo CPU that will be performing the open-loop and closed-loop control of the spindle. The servo CPUs are also numbered in ascending order from left to right. The module at the far left is No. 1.

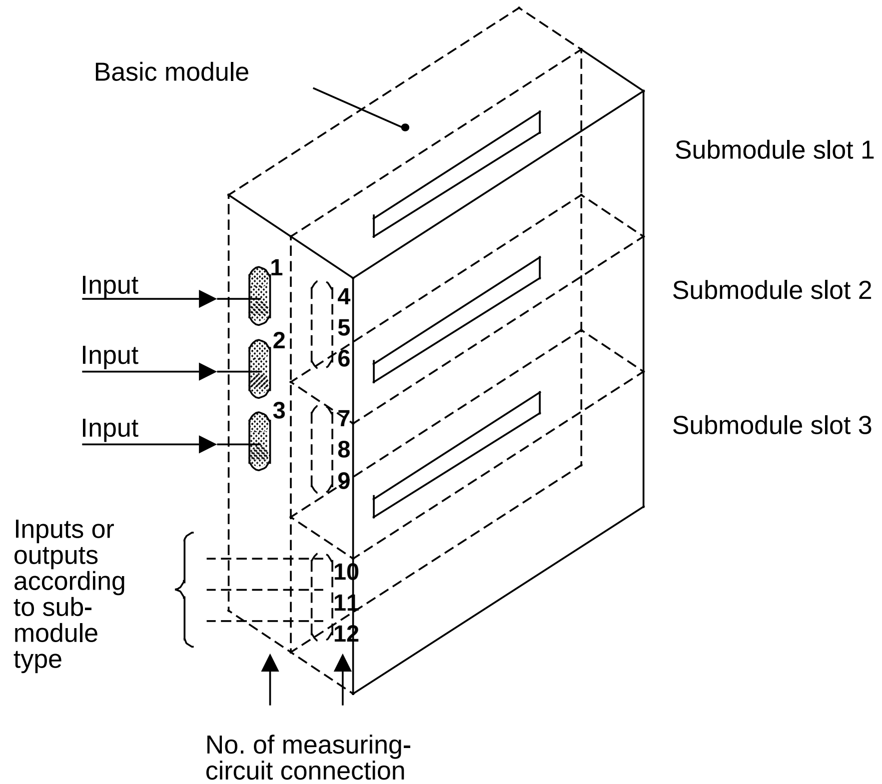
  Permitted values: 00 to 04

Notes

- The value "00 00 00 00" for MD 400* is only permitted if the spindle is not available for the control (MD 521*, bit 7 = 0).
- Both SPC and HMS measuring-circuit modules are permitted for actual value acquisition.



Assignment of inputs/outputs on SPC measuring-circuit modules



Assignment of inputs/outputs on HMS measuring-circuit modules (with setpoint submodules)

401*	**Drift compensation for spindle**		**401***
Standard value	Lower input limit	Upper input limit	Units
0	**- 500**	**500**	**VELO**

The value of this machine data must be altered until, with the same reference speed in both directions, the spindle runs at the same value of actual speed. The adjustments must be carried out at low speed.

The check can be made from the basic display, from the diagnostic display for the spindles or with a tachometer (for spindles without an encoder).

403*-410*	Maximum speed for 8 gear stages		403*-410*
Standard value	Lower input limit	Upper input limit	Units
see table	**0**	**99 999**	**rev/min or 0.1 rev/min**

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	403*	404*	405*	406*	407*	408*	409*	410*

The machine data establish the maximum spindle speeds which can be attained in the individual gear stages with maximum setpoint voltage.

In the case of gearboxes with less than 8 stages, the value 0 must be entered for the missing stages.

Standard values

Gear stage 1: 500
Gear stage 2: 1000
Gear stage 3: 4000
:
:
Gear stage 8: 4000

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min

411*-418*	Minimum speed for 8 gear stages		411*-418*
Standard value	Lower input limit	Upper input limit	Units
see table	**0**	**99 999**	**rev/min or 0.1 rev/min**

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	411*	412*	413*	414*	415*	416*	417*	418*

The minimum and maximum speeds (MD 403* and 410*) establish the speed ranges for the individual gear stages. From the programmed value of the spindle speed, therefore, it is possible to ascertain the necessary gear stage and signal it to the PLC. If the speed ranges overlap, the new gear stage must be chosen so that there are as few changes as possible.

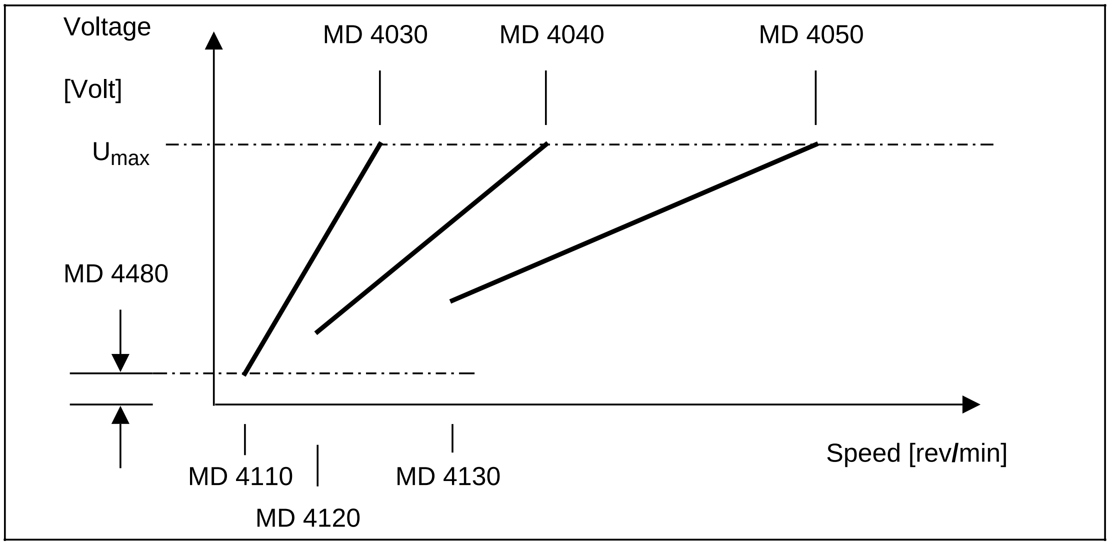
Standard values

Gear stage 1: 50
Gear stage 2: 500
Gear stage 3: 1000
Gear stage 4: 2000
:
:
Gear stage 8: 2000

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min



Speed ranges of the gear stages

U_{max}: max. spet speed voltage (see MD 468*)

419*-426*	**Acceleration time constant without position controller for 8 gear stages**		**419*-426***
Standard value	Lower input limit	Upper input limit	Units
1 000	**0**	**50 000**	**1 ms**

Active: When all channels of the mode group are in STOP state

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	419*	420*	421*	422*	423*	424*	425*	426*

The control issues the ramp-form setpoint (ramp-function generator) for acceleration and deceleration functions. The slope of the ramp is determined by the acceleration time constant.

Notes

- There are two acceleration time constants for each gear stage:
  - Acceleration time constant without position controller (MD 419* to 426*)
  - Acceleration time constant with position controller (MD 478* to 485*)

  Time constants without position controller are used for the open-loop control mode and time constants with position controller for the positioning mode and synchronous mode.
- MD 419* to 426* can be set so that the motor accelerates at maximum current for a limited time. On the other hand, there must always be a certain amount of control reserve with MD 478* to 485*. The current limit must never be reached even in the motor’s weak-field range.
- If there is already a ramp-function generator in the drive actuator, MD 419* to 426* can be set to with zero. There will then be a step-change in setpoint output for acceleration and deceleration functions.
- The acceleration value must be such that the speed is altered by more than half a revolution within one interpolator pulse (MD 155). Otherwise alarm 2291 is triggered.
- The time constants entered here are complied with only when selecting and deselecting while the spindle is running in the case of "oscillation". There is a step-change in setpoint output with direction reversal.

427*-434*	Creep speed for M19 for 8 gear stages		427*-434*
Standard value	Lower input limit	Upper input limit	Units
100	**0**	**16 000**	**rev/min or 0.1 rev/min**

Active: When all channels of the mode group are in STOP state

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	427*	428*	429*	430*	431*	432*	433*	434*

In the case of spindles on 32-bit CPUs, the creep speed serves as the limit which must not be exceeded during acceleration:

- When positioning from zero speed, maximum acceleration is to the creep speed.
- When the spindle is running, the position controller is activated when the creep speed is reached. The following then applies to the speed pattern:
  - If the actual speed is less than the creep speed, the spindle accelerates to the creep speed at the most.
  - The NC MDs 478 to 485 are active when braking from the creep speed to a standstill.

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min

435*-442*	Gain factor for 8 gear stages		435*-442*
Standard value	Lower input limit	Upper input limit	Units
833	**0, 166**	**16 000**	**0,01 s^{-1}**

Active: When all channels of the mode group are in STOP state

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	435*	436*	437*	438*	439*	440*	441*	442*

In the case of spindles the gain factor is used for the synchronous mode as well as for M19 (positioning mode).

Therefore, it is the same as a gain factor in a position control circuit.

Notes

- The "actual" gain factor of a gear stage used by a machine depends on several factors:
  - the gain factor setting for the gear stage
  - the maximum speed of the gear stage
  - the multgain factor (MD 468*)
  - the setting of speed controller and drive
- An input value of "0" opens the position control circuit.

443*	**Position tolerance for M19**		**443***
Standard value	Lower input limit	Upper input limit	Units
2 000	**0**	**16 000**	**units [MS]**

Active: When all channels of mode group in STOP state

In the positioning mode, IS: SPINDLE POSITION REACHED is output to the PLC as soon as the deviation from the set position is less than this tolerance.

This machine data has no direct effect on the accuracy of the positioning because the control system still attempts to reach the position as accurately as possible. It only ends when the positioning mode is aborted.

444*	**Tolerance of spindle speed**		**444***
Standard value	Lower input limit	Upper input limit	Units
10	**0**	**100**	**%**

Active: When all channels of the mode group in STOP state

The actual speed of the spindle is compared with the setpoint value. If the discrepancy is greater than the tolerance given here, IS: SPINDLE IN SET RANGE will be cancelled.

Notes

- The setpoint speed is determined by the programmed value of speed, and the speed override and the limits imposed by the speed limit values.
- An encoder must be mounted on the spindle for acquiring the actual value of speed. If there is no encoder, speed monitoring will not be possible.
- The value of tolerance entered is in the form of a percentage of the programmed value of setpoint speed.
- The permitted range is as follows:
  (setpoint speed - tolerance)   actual speed   (setpoint speed + tolerance)
- There is no monitoring if 100% is entered.

445*	Tolerance of maximum spindle speed		445*
Standard value	Lower input limit	Upper input limit	Units
10	**0**	**100**	**%**

Active: When all channels of the mode group in STOP state

If the speed limit for the spindle is exceeded by more than this tolerance, IS: SPEED LIMIT EXCEEDED is set and an alarm is triggered. All axes and spindles of the relevant mode group are stopped.

Notes

- An encoder must be mounted on the spindle for acquiring the actual speed. If not, it will be impossible to monitor the speed.
- The value of tolerance entered is in the form of a percentage of the current relevant speed limit.
- The current speed limit is determined by the smallest value of the following:
  - MD 403* to 410* "Max. speed per gear stage"
  - MD 451* "Max. chuck speed"
  - SD 401* "Programmable spindle speed limit for G96"; programmed with G92
  - SD 403* "Programmable spindle speed limit"; programmed with G26
- There is no monitoring if 100% is entered.

446*	Tolerance of zero speed		446*
Standard value	Lower input limit	Upper input limit	Units
10	**0**	**10 000**	**0.01 %**

Active: When all channels of the mode group are in STOP state

If the actual speed of the spindle is less than this tolerance, IS: SPINDLE STOPPED will be set.

Notes

- An encoder must be mounted on the spindle in order to acquire the actual value of speed. If not, it will be impossible to monitor the speed.
- The value of tolerance entered is in the form of a percentage (unit 0.01%) of the maximum speed of the current gear stage.
- There is no monitoring if 10 000 (100%) is entered.

447*	Delay for servo disable		447*
Standard value	Lower input limit	Upper input limit	Units
1 000	**0**	**16 000**	**ms**

Active: When all channels of the mode group are in STOP state

The following events will cause the enabling of the speed controller to be cancelled:

- Cancelling of IS: SERVO ENABLE
- EMERGENCY STOP
- Triggering of the measuring circuit monitoring
- Cancelling of IS: MODE GROUP READY
- In the synchronous spindle mode, spindle disable is issued.

However, the speed controller enable is only cancelled when the time given here has elapsed.

448*	Minimum motor speed setpoint		448*
Standard value	Lower input limit	Upper input limit	Units
50	**0**	**16 000**	**rev/min**

Active: When all channels of the mode group are in STOP state

The machine data establishes the minimum motor speed below which smooth running of the motor cannot be guaranteed.
The actual speed of the motor is never taken below this minimum even if, for a constant cutting rate (G96) and increasing turning diameter, it would be necessary, strictly speaking, to employ a lower speed. The end result, of course, is that the cutting rate is no longer constant.

449*	Basic speed		449*
Standard value	Lower input limit	Upper input limit	Units
50	**0**	**99 999**	**rev/min or 0.1 rev/min**

Active: When all channels of the mode group are in STOP state

When the PLC sets IS:PLC SPINDLE CONTROL and BASIC SPEED, the speed value entered here is output as the speed setpoint for the spindle. The current actual gear stage is taken into account.

Units

The units of the value entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min)

450*	Oscillation speed		450*
Standard value	Lower input limit	Upper input limit	Unit
20	**0**	**10 000**	**0.01%**

Active: When all channels of the mode group are in STOP state

When the PLC sets IS: PLC SPINDLE CONTROL and OSCILLATION SPEED, the value entered here is output as the speed setpoint for the motor, i. e. an input value of 10 000 corresponds to oscillation at maximum motor speed.

451*	Maximum spindle speed		451*
Standard value	Lower input limit	Upper input limit	Units
4 000	**0**	**99 999**	**rev/min or 0.1 rev/min**

Active: When all channels of the mode group are in STOP state

This machine data establishes the maximum speed for the spindle regardless of the gear stage. The set spindle speed is accordingly limited to this value.

If the speed is exceeded by more than the permitted tolerance (MD 445*), IS: SPEED LIMIT EXCEEDED is set and an alarm triggered.

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min

452*	Spindle position will external M19		452*
Standard value	Lower input limit	Upper input limit	Units
0	0	35 999	0.1° 0.01°

Active: After POWER ON

When the PLC sets IS: PLC SPINDLE CONTROL and POSITION SPINDLE, the spindle is driven to the angle entered here.

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit 0.1°
Bit 3 = 1: Unit 0.01°

453*	Spindle mode group		453*
Standard value	Lower input limit	Upper input limit	Units
1	1	8	–

Active: After warm restart

This machine data establishes to which mode group the spindle is assigned. The PLC can assign a spindle to a different mode group with the aid of the "warm restart" function.

455*	Pulses, variable incremental weighting		455*
Standard value	Lower input limit	Upper input limit	Units
32	**0**	**65 000**	**–**

456*	Distance traversed, variable incremental weighting		456*
Standard value	Lower input limit	Upper input limit	Units
5 625	**0**	**65 000**	**–**

Active: After POWER ON

The internal calculations for the spindle are performed with the value of resolution provided by the measuring system (i.e. the measuring system resolution). The measuring system resolution must be entered in order to perform the internal conversions with it. However, like the axes, the entry is not made by means of the increments of the encoder and the pulse multiplication, but by means of the variable incremental weighting. This allows spindle encoders of any pulse rate and also encoder gearboxes to be used.

The parameters for determining MD 455* and 456* are listed in the table below.

Parameter	**Symbol**	**Meaning**
Position control resolution (MD 524*)	b	Absolute unit of distance traversed; standard constant 0.5 x 10^{-3} degrees
Pulses per revolution	p	No. of encoder pulses per revolution
Pulse multiplication	f	Pulse multiplication for HMS measuring circuit module; MD 458*
Encoder gearbox	r1/r2	Gear ratio between spindle and encoder

MD 455* = $4 \cdot p \cdot f \cdot r2$ Number of measuring system pulses per encoder revolution including hardware multiplication; multiplied by r2

MD 456* = $\dfrac{360 \cdot r1}{b}$ Number of internally-calculated increments per encoder revolution; multiplied by r1

If the calculated values exceed the entry limits, common factors **must** be abbreviated.

Notes

- The position control resolution for the spindle has been set with NC MD 524*,.
- If a C axis is assigned to the spindle (MD 461*), the C axis resolution is used for calculation (MD 1800*, 364*, 368*).
- The standard values are applicable to the use of an SPC measuring-circuit module and an encoder with 1024 pulses per revolution (see example).
- In the C axis mode the increment weighing of the C axis is used for the display of the spindle revolutions.

Examples

- Encoder mounted directly on the first spindle; square-wave encoder and SPC measuring-circuit module
  b = 0.5×10^{-3} degrees
  p = 1024 pulses per revolution
  f = 1 no pulse multiplication
  r1 = 1 no encoder gearbox
  r2 = 1

  The encoder delivers 4 x 1024 = 4096 pulses when the spindle performs one complete revolution.

  When these factors are substituted in the formulae:

  MD 4550 = 4 096
  MD 4560 = 720 000

  Abbreviating by means of the common factor 128 gives:

  MD 4550 = 32
  MD 4560 = 5625

- Encoder mounted on the second spindle through an encoder gearbox; SIPOS unconditioned signal generator and HMS measuring-circuit module
  b = 0.5×10^{-3} degrees
  p = 2500 pulses per revolution
  f = 8 pulse multiplication by HMS
  r1 = 2 2 spindle revolutions correspond to
  r2 = 3 3 encoder revolutions

  When these factors are substituted in the formulae:

  MD 4550 = 240 000
  MD 4560 = 1 444 000

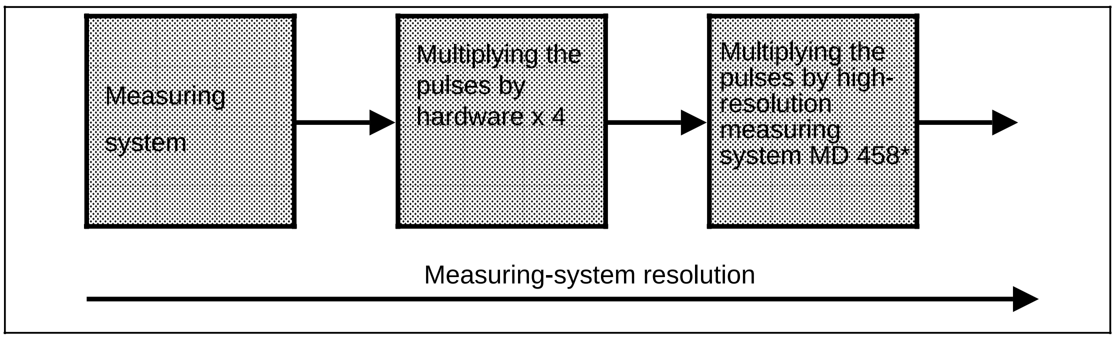
  Abbreviating by means of the common factor 240 000 gives:

  MD 4550 = 1
  MD 4560 = 6

458*	Pulse multiplication by HMS		458*
Standard values	Lower input limit	Upper input limit	Units
1	**0**	**128**	**–**

Active: After POWER ON

With the HMS measuring-circuit module it is possible to adapt the resolution of the measuring system for every type of application. The pulses coming from the encoder are multiplied in two stages:



Pulse multiplication by HMS

The following values are permitted for MD 458* when an HMS measuring-circuit module is used:

1, 2, 4, 8, 16, 32, 64 und 128

Overall, therefore, pulse multiplication by the factor 512 can be obtained.

Notes

- The machine data is only taken into account when an HMS measuring-circuit module is used.
- It is essential for the multiplication factor to be taken into account with variable incremental weighting (MD 455*, 456*).

459*	Shift of the zero mark		459*
Standard value	Lower input limit	Upper input limit	Units
0	**-36 000**	**36 000**	**0.01°**

Active: After POWER ON

The relocatable zero mark enables the angle zero of the spindle (e.g. to M19 S0) to differ from the zero mark of the encoder. It allows the spindle zero to be set anywhere.

The reference system of the control for the spindle is shifted in relation to the reference system of the encoder.

Notes

- Any change in this machine data only becomes operative when the spindle has been resynchronized with the encoder.
- When a positive value is issued to MD 459*, the angular position of the spindle is shifted in the direction corresponding to clockwise rotation (M03).
- In the case of spindle/C axis combination, the zero mark shifts NC MD 244* and 459* must be identical.

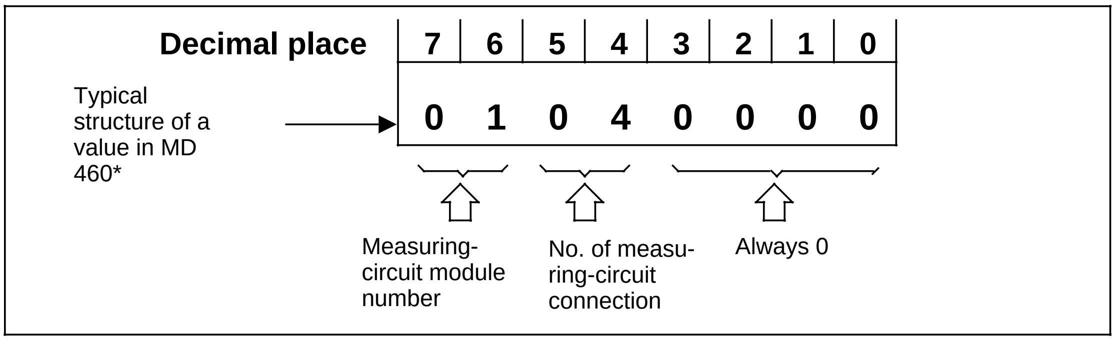
Example

MD 4590 = 9 000	Shift of zero mark by 90°
M19 S270 L_F	The spindle is positioned at 270° in the control reference system; since this system has been shifted by 90° compared with the zero mark of the encoder, the spindle will stop at the zero mark of the encoder
MD 2440 = 90 000	With a positon control resolution of $0.5 \cdot 10^{-3}$ mm.

460*	Spindle assignment, setpoint output		460*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**see Permitted values**	**–**

Active: After POWER ON

MD 460* establishes the assignment of the analog speed setpoint to a particular measuring-circuit module and output.



*Structure of MD 460**

Meanings of terms

- The **measuring-circuit module number** is the number of the module on theservo local bus. The modules are numbered from left to right in ascending order. The module at the far left has the number 1.
- The **number of the measuring-circuit connection** refers to the number of the output on the selected HMS or SPC module. The numbering of the inputs and outputs is shown in the appropriate figure.

Permitted values:	04 to 06	for SPC modules
	04 to 12	for HMS modules

Note

- The value "00 00 00 00" for MD 460* is only permitted if the spindle is not available for control (MD 521*, bit 7 = 0).

461*	Global axis No. of assigned C axis		461*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**24**	**–**

Active: After POWER ON

This machine data establishes the global axis number under which a spindle will be operated when it is working as a C axis. It determines, for example, which axis-specific machine data record will be used.

The machine data is also used when assigning a following spindle to its leading spindle (see MD 9x0, x = 4, 6, 8 for FS 1, 2, 3).

If the value 0 is entered in MD 461*, the spindle can be used neither as a C axis nor as a following spindle.

Notes

- The C axis must be defined on the same servo CPU as the spindle (MD 200* and 400*).
- If a C axis is assigned to a spindle (MD 461* not 0), the internal calculations for the spindle are also performed using the measuring-system resolution for the C axis. Therefore, the variable incremental weighting of the axis must also be entered (MD 364*, 368*, 1800*).
- If a global C-axis number is assigned to a spindle with machine data MD 461*, this C axis can only be traversed when it is selected as a C axis. Selection/deselection is via M functions from MD 260/MD 261.

  Example:

  MD 260=70, MD 261=71, MD 4610=2 (C axis)

  C axis cannot be traversed until it has been selected with M70.

  C axis operation is deselected with M71.
- Refer to Function Manual "Extended Spindle Functions" for further information on the C axis.

462*	Cutoff frequency of spindle encoder		462*
Standard value	Lower input limit	Upper input limit	Units
500	**0**	**16 000**	**kHz**

Active: After POWER ON

The cutoff frequency of the actual-value encoder of the spindle is entered in the machine data. The value can be taken from the manufacturer's documentation.

It is possible for pulses from the encoder to be lost if the cutoff frequency is exceeded. It means that the actual-value acquisition will be incorrect and IS: SPINDLE SYNCHRONIZED will be cancelled.

When the value falls below the critical frequency again, the spindle is automatically re-synchronized with the encoder (hysteresis characteristic).

Note

If there is no separate encoder for the C axis mode, the cutoff frequency of the spindle encoder will be used for monitoring this mode too.

463*	Synchronous position shift		463*
Standard value	Lower input limit	Upper input limit	Unit
0	**-17 999**	**18 000**	**0.01°**

Active: When all channels of the mode group are in STOP state

A specific value of angular offset between leading spindle and following spindle can be programmed when the synchronous mode is selected.

For example, if an angular offset of 0 degrees is programmed with a speed ratio of 1:1 (G201 S1 = A0 L_F) it means that the following spindle will always reach its zero mark when the leading spindle reaches its. The respective values of zero mark shift (MD 459*) are taken into account.

With MD 463* it is possible to assign parameters for an angular offset between leading spindle and following spindle which will be allowed for as standard when the synchronous mode with a defined angular offset is selected. If an angle of 0 degrees is then programmed with G201, the value of angular offset entered in MD 463* will be taken.

Notes

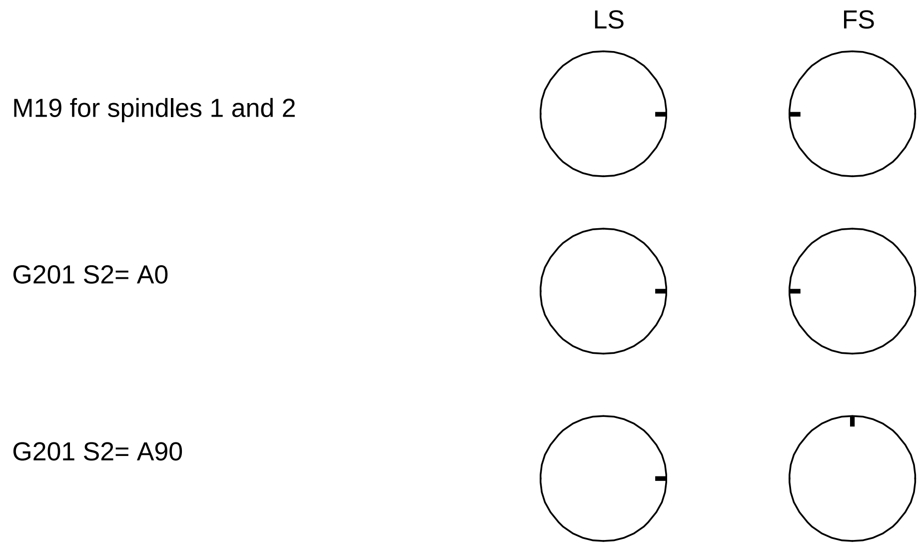
- Synchronous position shift must be entered for the following spindle. The machine data of the leading spindle is not taken into account.
- When issuing a positive value to MD 459*, the synchronous position of the following spindle is shifted in the direction corresponding to clockwise rotation (M03).

Examples

Spindle 1:	Leading spindle
Spindle 2:	Following spindle for spindle 1 (speed ratio 1:1)
Direction:	Both spindles rotate clockwise with M03
M19 position:	SD 4020 = 0 SD 4021 = 0

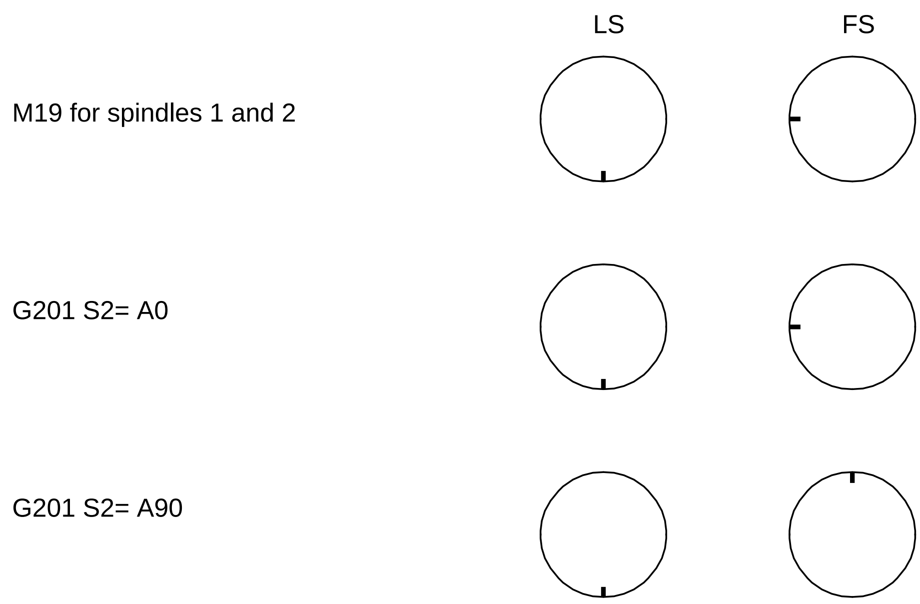
- MD 4590 = 0
  MD 4591 = 0
  MD 4631 = 0

  i.e. positive shift of zero mark and synchronous position



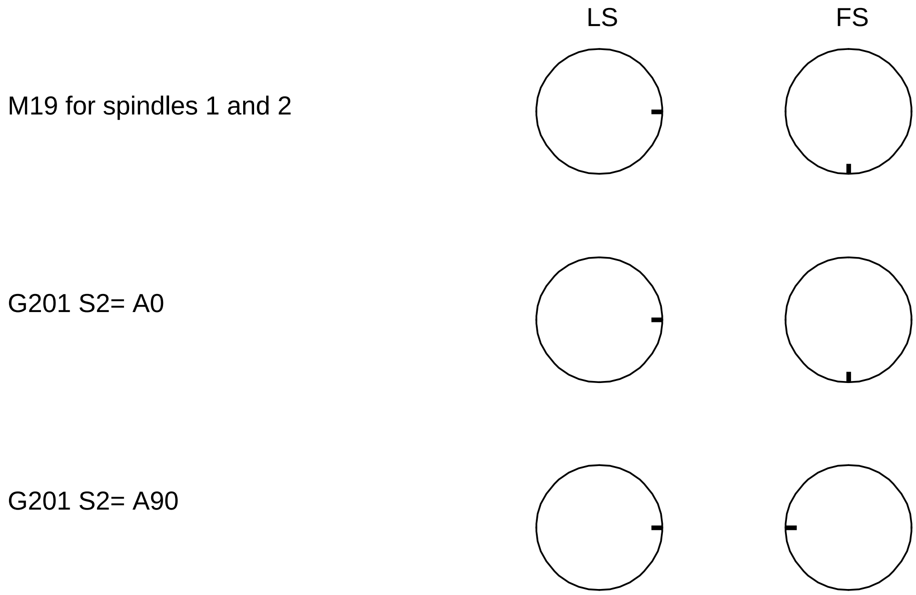
- MD 4590 = 9 000
  MD 4591 = 0
  MD 4631 = 0

  i.e. positive shift of zero mark of LS; no shift of zero mark of FS and synchronous position



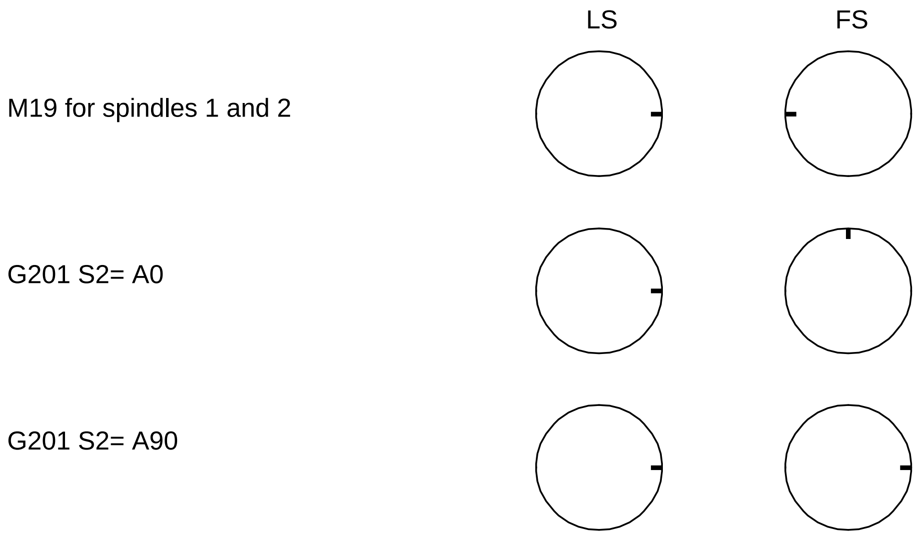
- MD 4590 = 0
  MD 4591 = – 9 000
  MD 4631 = 0

  i.e. negative shift of zero mark of FS; no shift of zero mark of LS and synchronous position



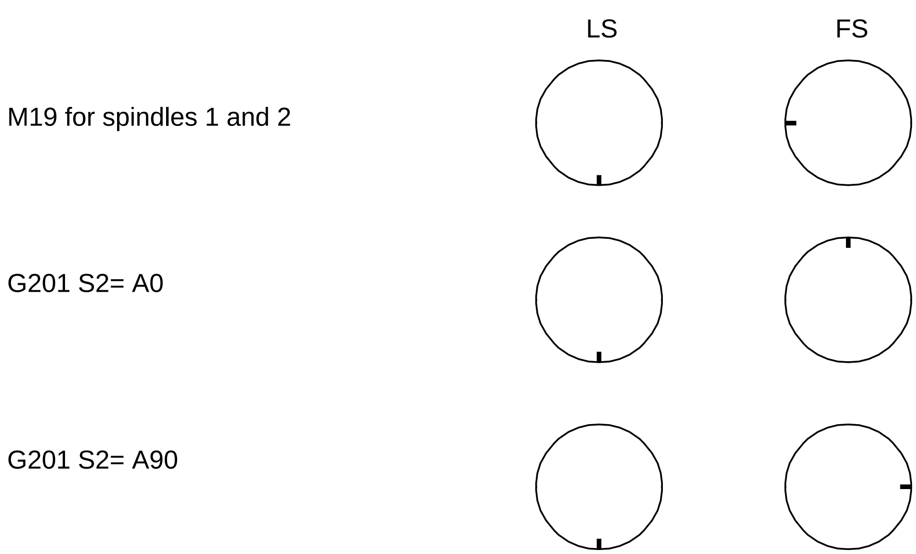
- MD 4590 = 0
  MD 4591 = 0
  MD 4631 = 9 000

  i.e. no shift of zero marks; positive shift of synchronous position



- MD 4590 = 9 000
  MD 4591 = 0
  MD 4631 = 9 000

  i.e. positive shift of zero mark of LS and synchronous position; no shift of zero mark of FS



Application

Through MD 463* it is possible to compensate for any offset of the zero marks of the leading spindle and following spindle without having to shift the zero marks of the individual spindles.

465*	P-component feedforward control factor		465*
Standard value	Lower input limit	Upper input limit	Unit
0	0	1 000	0.1 %

Static feedforward control is activated by entering a value other than zero in MD 465*. With position-controlled spindles (positioning mode, C axis mode, synchronous mode) it allows the following error to be reduced and the dynamic response to be improved.

Entering the value 1000 produces driving at constant speed with no following error. With this setting, however, some overshoot of the preset position or travel contour must be anticipated during acceleration and deceleration functions.

Entering a value other than zero in MD 467* too (time constant for feedforward control), activates dynamic feedforward control, which improves the dynamic behaviour of the spindle.

Dynamic feedforward control should be activated in order to obtain precise synchronism in the synchronous mode. The feedforward control settings must be the same for both spindles.

Recommended range of values

Machine tests have shown that the best results are obtained with values in the range from 500 to 800.

Note:

The "feedforward control" Option must be active.

466*	Ratio of position control cycle to servo cycle time		466*
Standard value	Lower input limit	Upper input limit	Unit
1	1	64	–

Active: After POWER ON

In the basic setting, all the spindles of a servo CPU have a set position control cycle that corresponds to the servo cycle time. The servo cycle time is calculated as follows:

Servo cycle time = System clock . MD 16x Possible inputs; 1, 2, 4, 8, 16, 32, 64:

x = 0 for servo CPU 1
1 for servo CPU 2
2 for servo CPU 3
3 for servo CPU 4

The position control cycle can set spindle-specifically with MD 466* as follows:

Position control cycle = Servo cycle time · MD 466*

$$\text{Condition:} \quad \frac{\text{Interpolation time:}}{\text{Posit. control cycle}} = \text{Integer}$$

467*	Time constant for dynamic feedforward control		467*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**16 000**	**0.1 ms**

The leading spindle and following spindle must have identical dynamic response for command behaviour.

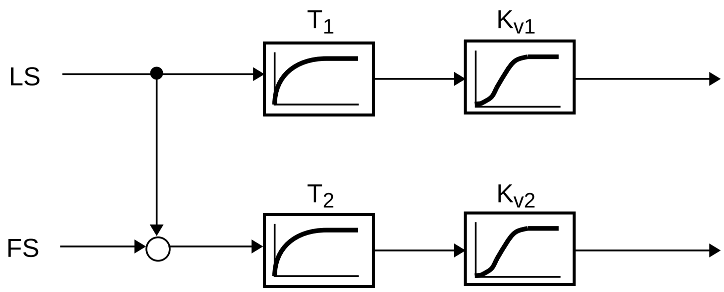
- Matching the dynamic response with identical servo gain (K_V) factors.
  The values of position controller gain for the leading spindle and following spindle should be identical if possible and should also be as high as possible in order to obtain good disturbance characteristics.
- If the leading spindle and following spindle cannot be matched for the same dynamic response, the faster spindle can be slowed down by means of a delay element in the setpoint branch for command behaviour (NC MD 486*).

Setting instructions

- Set the individual spindles for good disturbance characteristics and command behaviour.
- Calculate the equivalent time constants for the individual spindles from the formula:

$$T_{equiv} = \frac{1}{K_V}$$

- It is a precondition that the measured values of following error be correct (set maximum speed and multgain accordingly).
- Set the spindle-specific delay element (MD 467*) for the faster spindle by entering the difference between the equivalent time constants of the two spindles as the time constant.
- Fine-balance by selecting the synchronous mode (without compensatory controller) and altering the delay time constant at constant speed until the two values of following error are the same.



$K_{v1} = 1000 \dfrac{1/\text{min}}{360°} = \dfrac{1000}{60} \; S^{-1}$ $\quad T_{equiv1} = 60$ ms

$K_{v2} = 2000 \dfrac{1/\text{min}}{360°} = \dfrac{2000}{60} \; S^{-1}$ $\quad T_{equiv2} = 30$ ms

$T_1 = 60$ ms - 60 ms = 0 ms

$T_2 = 60$ ms - 30 ms = 30 ms

- In order to save computing time, one delay element should always have a time constant of zero.

468*	Multgain factor		468*
Standard value	Lower input limit	Upper input limit	Units
9 000	**0**	**32 000**	**mV**

Active: When all channels of the mode group are in STOP state

Like the axis-specific multgain factor (MD 260*), the spindle-specific value is used for digital tachogenerator matching.

Assuming that U_{max} is the set speed voltage at maximum motor speed, then:

Multgain = U_{max} [mV]

Example

U_{max}	=	8.345 V
Multgain	=	8 345

Special features with synchronous operation:

Since the leading and following spindle operate with position control in the synchronous mode, the multgain must be set so that a control margin is available. The control margin should be 5 %.

Example:

Maximum speed of spindle with 10 V set speed voltage: 2000 rpm
2000 V/min

5 % control margin = approx. 0.5 V set speed voltage = approx. 100 rpm

Maximum speed (MD 403*)	:	1900 rpm
Multgain factor (MD 468*)	:	9500

If 1900 rpm is programmed, a set speed voltage of 9.5 V is output.

469*	**Factor for gain change**		**469***
Standard value	Lower input limit	Upper input limit	Units
100	**1**	**16 000**	**%**

In the positioning mode, the spindle must be held in the target position under position control. In order to ensure a steady stationary position for the spindle even when there is drift, very small speed setpoints with a very high resolution must be fed to the drive actuator through the analog interface.

Therefore, with some drive actuators it is possible, by applying a configurable terminal signal, to select different normalization for the speed setpoints.

This changing of the normalization in the drive actuator must be taken into account in the control system so that, overall, the effective gain factor remains the same. The gain factor (dependent on gear stage, MD 435* to 442*) must be matched to the new normalization .

The factor for gain changing is then:

$$\text{MD 469*} = \frac{1}{N} \quad \text{where N = normalization factor in the drive actuator}$$

Notes

- Changing the gain factor is initiated by setting IS: CHANGE GAIN FACTOR. This signal must always be set simultaneously with the terminal signal.
- Entering the value 0 in MD 469* (the standard value) is the same as entering the value 100. In both cases the gain factor given in MD 435* to 442* is not changed.

478*-485*	Acceleration time constant with position controller for 8 gear stages		478*-485*
Standard value	Lower input limit	Upper input limit	Units
3 000	**0**	**50 000**	**1 ms**

Active: When all channels of the mode group are in STOP state

Assignment

Gear stage	1	2	3	4	5	6	7	8
NC MD	478*	479*	480*	481*	482*	483*	484*	485*

For acceleration and deceleration, the control system issues ramp-form setpoints (ramp-function generator). The slope of the ramp is determined by the acceleration time constant.

Notes

- There are two acceleration time constants for each gear stage:
  - Acceleration time constant without position controller (MD 419* to 426*)
  - Acceleration time constant with position controller (MD 478* to 485*)

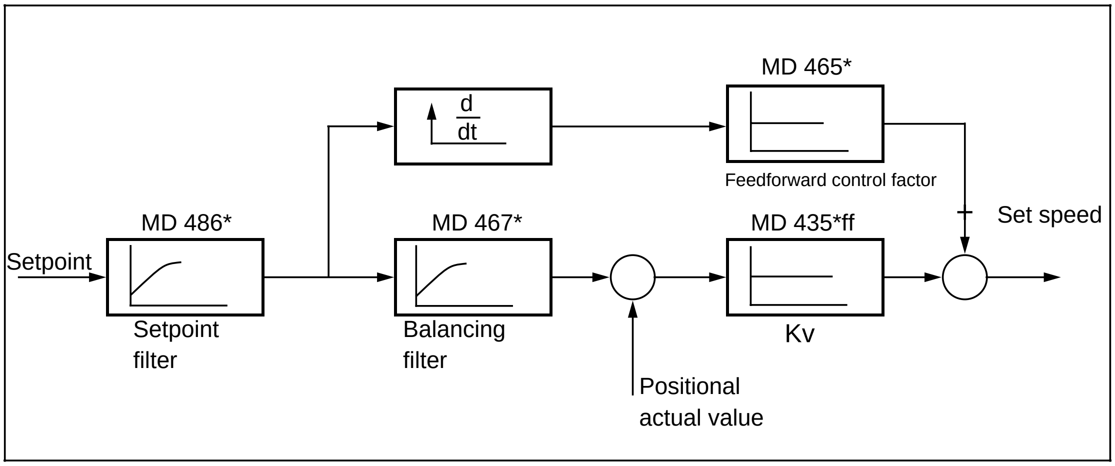
  In the open-loop control mode, time constants without position controller are used; in the positioning mode and synchronous mode, time constants with position controller.
- MD 478* to 485* must be set so that the motor can follow the setpoint at all times without impinging on the current limit. This is particularly important in the field-weakening range of the motor.
- The acceleration value must be such that the speed is altered by more than half a revolution within one interpolator pulse (MD 155). Otherwise alarm 2291 is triggered.

486*	Time constant setpoint smoothing	486*

Standard value	Lower input limit	Upper input limit	Units
0	**0**	**16 000**	**0.1 ms**

Active: Immediately

The setpoint filter prevents overshooting of the spindle position with dynamic speed feedforward control (NC MD 465*) of 100 %.



Setpoint smoothing does not have to be selected by an additional activation bit as it does with the axes.

9.3.5.4 Machine data bits

Allocation

NC MD	Meaning
5000 : 5199	**General bits**
520* : 522*	**Spindle-specific bits (up to 6 spindles)**
540* : 558*	**Channel-specific bits (up to 8 (out of 16) channels)**
560* : 596*	**Axis-specific bits (max. 24 axes)**
6000 : 6999	**Leadscrew error compensation bits**
1800* : 1804*	**Axis-specific bits (max. 24 axes)**

The symbol "*" in channel, axis and spindle-specific MD numbers stands for the channel/axis/spindle designation.

You can substitute the following values for "*":

0	1st axis or 1st channel or 1st spindle
1	2nd axis or 2nd channel or 2nd spindle
2	3rd axis or 3rd channel or 3rd spindle
3	4th axis or 4th channel or 4th spindle
:	:
7	8th axis or 8th channel
:	:
11	12th axis
:	:
23	24th axis

The axis names (X, Y, Z, ...) are assigned using axis-specific NC MD bit 568*.

Notes:

The significance of the individual NC MD bits always refers to the **set** bit.
If a bit is **not** set, the declaration (name) is to be negated.
With SINUMERIK 880 GA2, the user can activate only 8 of the available 16 channels.

5000			NC MD Bit No.				5000
7	6	5	4	3	2	1	0
				Name of radius and chamfer			

Active: In next block

Name of **radius** and **chamfer** for:

- Contour definition
- Circular-path programming
- Polar coordinates

3	2	1	0

Bit No.

Address name

Key:

X = 0000
Y = 0001
Z = 0010
A = 0011
B = 0100
C = 0101
U = 0110
V = 0111
W = 1000
Q = 1001
E = 1010

Examples: 0100=B
0110=U

The names in MD 5000, MD 5001 and 568* must not coincide. The same axis name with a different extended address is not regarded as coinciding.

Permissible names for axes, angles, chamfer and radius

A	*Unassigned address*	N	Subblock
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	Feed	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*

5001		NC MD Bit No.					5001
7	6	5	4	3	2	1	0
				Name angle			

Active: In next block

3	2	1	0

Bit No.

Address name

Key:

X = 0000
Y = 0001
Z = 0010
A = 0011
B = 0100
C = 0101
U = 0110
V = 0111
W = 1000
Q = 1001
E = 1010

Example: 0011 = A

The names in MD 5000, MD 5001 and 568* must not coincide. The same axis name with a different extended address is not regarded as coinciding.

Permissible names for axes, angles, chamfer and radius

A	*Unassigned address*	N	Subblock
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	Feed	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*

5002	NC-MD						5002
Bit No.							
7	6	5	4	3	2	1	0
Input resolution				0	0	0	0

Active: After POWER ON

The **input resolution** (IS) specifies the increment weighting for dimensional inputs and displays. At the same time the reset position G70 (inch) or G71 (mm) is defined.

MD 5002, input resolution

metric

Bit	7	6	5	4	mm/degrees
	--	--	--	--	10^{-1}
	1	0	0	0	10^{-2}
	0	1	0	0	10^{-3}
	0	0	1	0	10^{-4}
	1	0	1	0	10^{-5}
	--	--	--	--	10^{-6}

Bit	7	6	5	4	inch/degrees
	--	--	--	--	10^{-1}
	--	--	--	--	10^{-2}
	0	1	0	1	10^{-3}
	1	1	0	1	10^{-4}
	1	0	1	1	10^{-5}
	0	1	1	1	10^{-6}

All linear axes **and** rotary axes of the control can be predefined with an axis-specific position control and display resolution. The input resolution and the geometry resolution resulting from it (responsible for programming, TO, ZO, interpolation, ...) is defined once for the whole control.

With these so-called axis-specific resolutions (for a more exact description see Section 11.7) it is possible to make optimum adaptations to the machine as regards.

- max. speed
- different accuracy
- max. traversing range.

Input resolution NC MD 5002 bit 4 to 7
Axis-specific display resolution NC MD 1800* bit 4 to 7
Axis-specific position control resolution NC MD 1800* bit 0 to 3

General bits

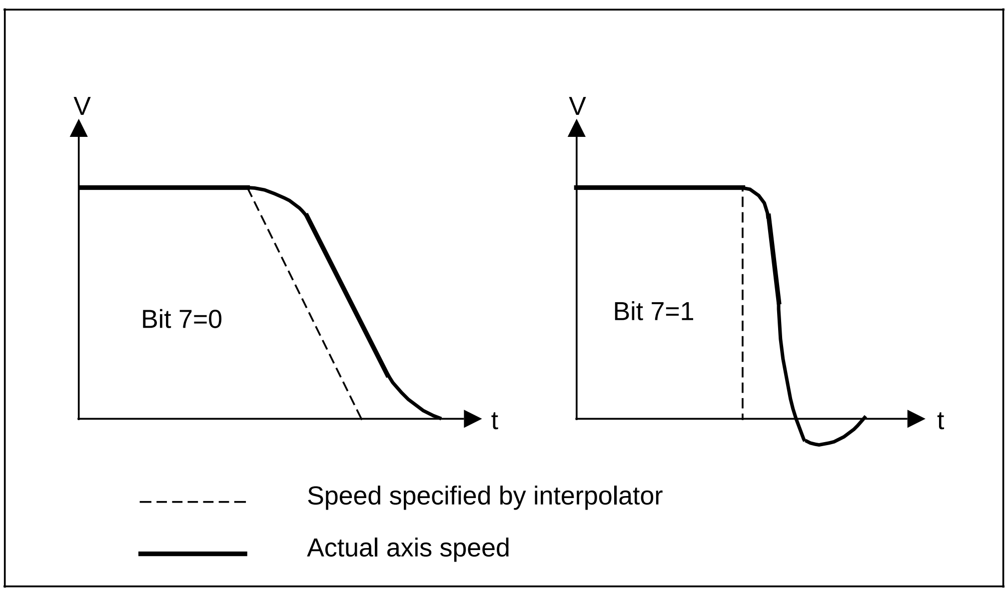
5003	NC MD Bit No.						5003
7	6	5	4	3	2	1	0
No deceleration at limit switch	JOG working area limitation active	Interpol. paramet. dep. on G91/G90	Angle for polar coord. dep. on G91/G90	Do not clear PRESET OFFSET with POWER ON	Auxiliary function output before travel	Channel-spanning overstore	Channel-spanning overstore

Bit 7: No deceleration on reaching the software limit switch.
With the bit set, braking is not performed with the acceleration/deceleration characteristic, only the following error is suppressed. The limit switch is not overshot as far (also see NC MD 1100*).

Active: Immediately

Note:

In the case of interpolating axes, all axes are shut down when the range limit of one axis has been reached. However, stopping without contour violation is only guaranteed when NC MD No. 5003 bit 7 (no deceleration at limit switch) has not been set, i.e. during braking via the acceleration ramp.



Bit 6: Working area limitation is also active in manual modes JOG, INC, REPOS. In addition to the software limit switches, the working area may now also be limited in JOG mode, thus ensuring that the machine is better safeguarded against unintentional traversing. However, since working area limitation is also a software limitation, proper operation cannot be ensured until after reference point approach.

Active: With mode change

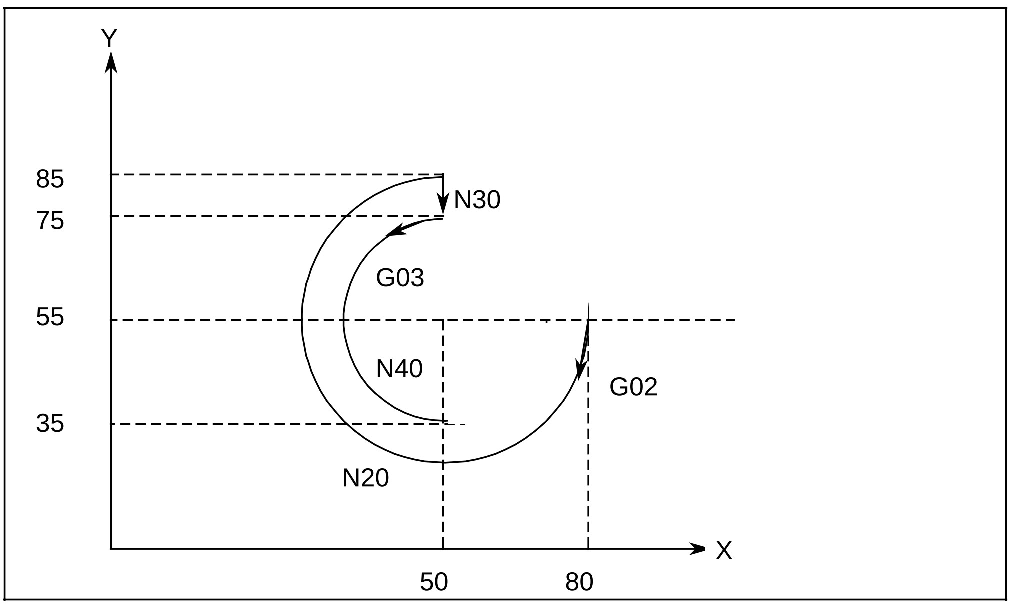
Bit 5: If bit 5 is set, the interpolation parameters (I, J, K) can be programmed in the block either absolutely (G90) or incrementally (G91) (also see NC MD 5007 bit 5). The interpolation parameters for contour definitions (blueprint programming sprint 1) must always be defined incrementally (G91) independently of bit 5.

Active: In next block

Example:

N10	G0	G90	X80	Y55	F500	LF	Approach in rapid traverse
N20	G02	X50	Y85	I50	J55	LF	Arc
N30	G01	X50	Y75	LF			Linear movement
N40	G03	X50	Y35	I50	J55	LF	Arc
N50	M30	LF					

If the interpolation parameters are specified absolutely, the centre refers to the workpiece centre and not to the circle initial point.



If an interpolation parameter is not programmed because it is "0", the G function which is located before the programmed interpolation parameter also refers to the unprogrammed interpolation parameter.

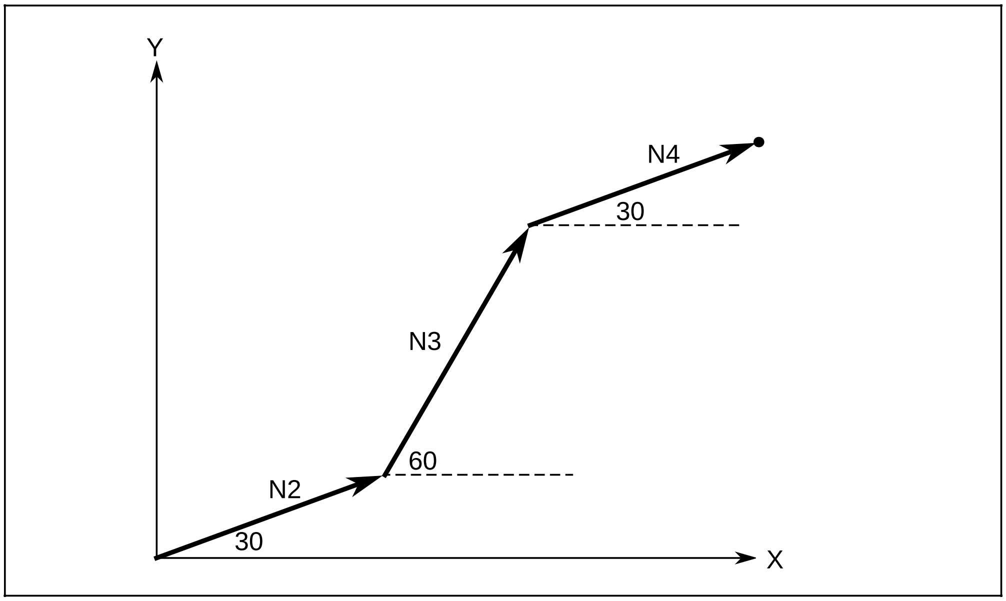
Bit 4: If bit 4 is set, angles in polar coordinate programming (G10, G11, G12, G13, G110, G111) can be programmed in the block either absolutely (G90) or incrementally (G91) (also see NC MD 5007 bit 5).

Active: In next block

Example:

N10 G0 G90 X0 Y0 LF

N20 G11 X0 Y0 A30 B40 F500 LF
N30 G91 G110 A60 LF
N40 G90 G110 A30 B30 LF
N50 M30 LF



Bit 3: Bit 3 = 1: The old "PRESET OFFSET" is assumed automatically once more after approach to the reference point.

Active: Immediately

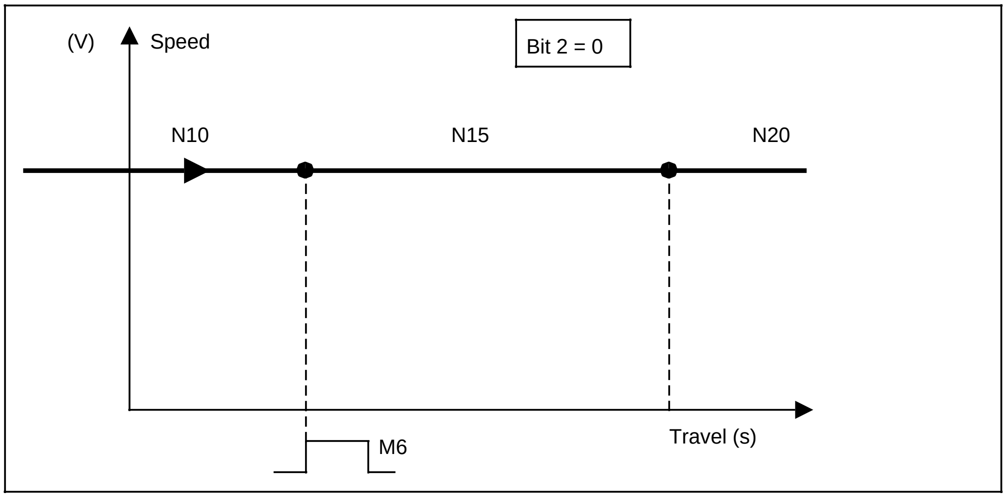
Bit 2: Active: Immediately

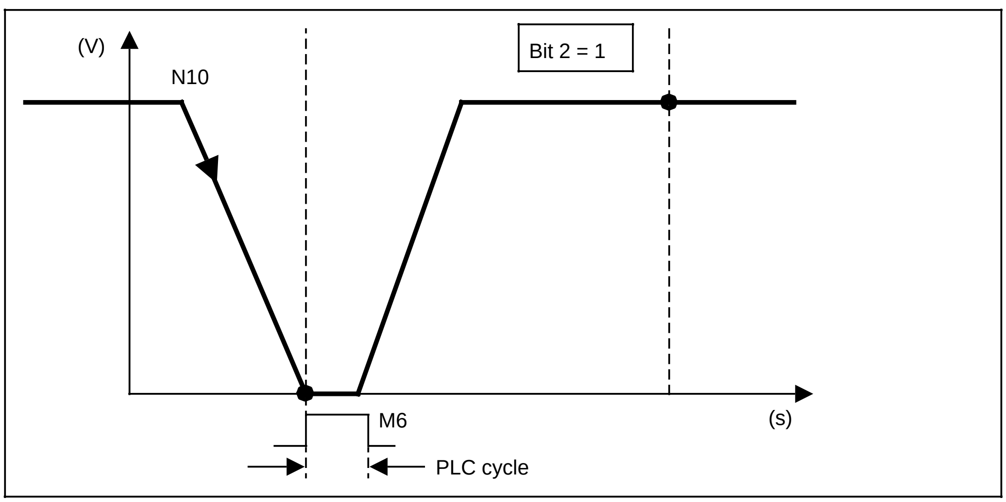
Bit 2 = 0, auxiliary function output during travel
Bit 2 = 1, auxiliary function output before travel
If an auxiliary function (M, S, T, H, D) is also programmed in a traverse block, the following two states may occur with the aid of **bit 2**:

Example: N 10
N 15 M6 X100 LF

N 20

With MD 5003, bit 2, functions G62/G64 can no longer be processed!





Bit 1: Bit 1 = 1 If the function overstore is triggered in a channel in which "Read-in disable" or "NC Stop" is active, the auxiliary functions can be corrected or deleted at any time and overstore can be triggered in other channels.

Bit 1 = 0 Overstore logic as usual.
If the function overstore is triggered in a channel in which "Read-in disable" or "NC Stop" is active, the auxiliary functions can no longer be corrected or deleted and overstore is not possible in other channels. The message "Overstore still active" is displayed.

5004	NC MD						5004
	Bit No.						
7	6	5	4	3	2	1	0
	Mode group-specific single block type B	Mode group-specific single block type A	Own rapid traverse override	NC-Start without ref. point	3rd handwheel available	2nd handwheel available	1st handwheel available

All bits active: Immediately

Bit 6: The first channel to have processed a block brings the other channels in the same mode group to a stop. When next started, the stopped axes continue to operate at the stop point. The channel triggering the stop starts with the next block. Single block type B.

Bit 5: The next NC start is only active if all channels in the same mode group have processed their block and if the axes are stationary. Single block type A.

Bit 4: The rapid traverse override switch usually featuring 4 positions (MD 147 to150) is transmitted by the PLC to the interface, where it is then sampled by the NC with bit 4 = 1. It is only active with G00.

Bit 3: The "NC START" PLC signal (DB10-17 DR2 bit 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. In this case MD 560* bit 4 has no significance. In order to ensure correct workpiece machining, synchronization must be performed otherwise, e.g. using the scratch method.

Bit 2,1,0: The number of available handwheels is specified, e.g.

Bit 2	Bit 1	Bit 0	Meaning
0	0	1	1 handwheel
0	1	1	2 handwheels
1	1	1	3 handwheels

5005	NC MD						5005
Bit No.							
7	6	5	4	3	2	1	0
Keyswitch group (configurable)			Keyswitch active for:				Reserved
			ZO fine	ZO coarse R parameter Angle of rotation	TO wear	TO geometry, basic	

Active: Immediately

Bit 7-5: Used to inhibit configurable data records via keyswitch (also see WS 800 Description).

Bit 4-0: If the corresponding NC MD bits are set, input and modification via the NC operator panel can be locked with the keyswitch:

Bit 4: Zero offset (G54-57) , fine
Bit 3: R parameter, angle of rotation, ZO (G54 to G57), coarse
Bit 2: TO wear (parameter P5 to P7)
Bit 1: TO geometry (parameter P2 to P4) and TO basic (parameter P8 and P9)

Bits 2 and 1 are only active if the standard display for tool offsets (P0 to P9) is not altered. When configuring tool offsets, the keyswitch group (bits 5 to 7) for the data fields (P0 to P_{xy}) must also be configured with the NC workstation (WS 800).

5006		NC MD					5006
			Bit No.				
7	6	5	4	3	2	1	0
Keyswitch active for:							
Teleservice	Axis converter	Selection Initial reset mode	TEACH IN PLAY-BACK	PP manual input and PP handling	Dry Run	DRF	OVER-STORE

Active: Immediately

Bit 7-0: If the corresponding NC MD bits are set, input and modification via the NC operator panel can be locked with the keyswitch:

Bit 7: Selection of Teleservice

Bit 6: Modification of axis addresses for axis converter

Bit 5: It is no longer possible to select initial reset mode when NC MD 5006, bit 5 (keyswitch active with initial reset) is set and keyswitch is in "Off" position.

Bit 4: TEACH-IN, PLAYBACK function (also see Operator's Guide)

Bit 3: For input of part programs via operator panel (EDIT, INPUT, CANCEL) and for PP-handling (COPY, RENAME, DELETE)

Bit 2: For selecting dry run feedrate

Bit 1: For traversing via handwheel in automatic mode (DRF function)

Bit 0: For transferring (overstoring) H, S, M, T, D functions in OVERSTORE submode

5007	NC MD						5007
	Bit No.						
7	6	5	4	3	2	1	0
TO in diameter	TO wear not active	Mixed programming G90/91 in block		Basic tool dimension active	No output of M17	G53 as for @706	Length compens. also with non-prog axes

Active: In next block

Bit 7: **Bit 7 = 0** Cutter (tool parameter P1 = 20) is defined as length (P2) and **radius** (P4).

Bit 7 = 1 Cutter is defined as length and **diameter**.
Cutter radius for a turning tool is always a radius.

Bit 7 affects P4 and P7 for 20...39. Bit 7 = 1 is also effective with P1 = 0 if NC MD 5008 bit 4 = 1.

Bit 6: All tool wear data can be declared invalid. The tool wear data (P5 to P7) can then be used for any user-specific purpose.
A value can then be added to or subtracted from the tool geometry data using the ”EDIT” key.

Bit 5: In a part program you always define for a whole block (up to LF) if the programmed values are to be interpreted as absolute (G90) or incremental (G91). There are numerous exceptions to this rule, for example scaling a circle centre, which must always be specified as incremental (G91) referring to the circle's initial point.

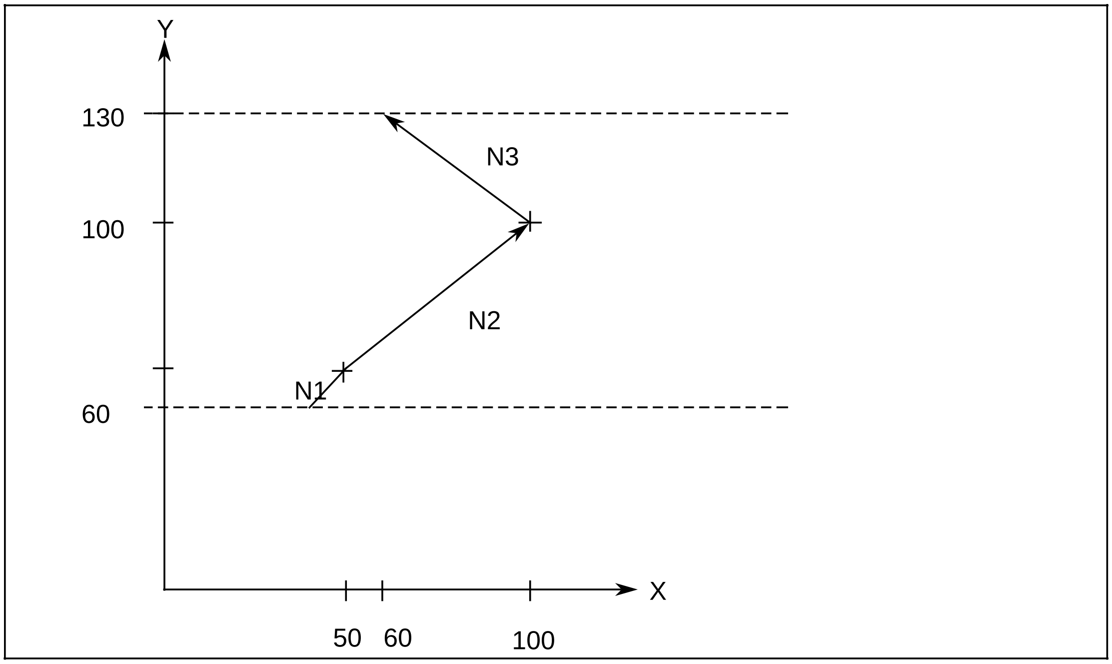
If bit 5 = 1, you can define in front of every program value (axis, interpolation parameter, angle) whether the value is to be interpreted as absolute (G90) or incremental (G91). (For interpolation parameters and angles also see NC MD 5003, bit 4 and bit 5).

	Axes	Interpol.Par.	Angles
Machine data 5003 bit4, 5=0 G90	ABS	ABS	ABS
Machine data 5003 bit4, 5=0 G91	INK	INK	INK
Machine data 5003 bit4=1 G90	ABS	INK	ABS
Machine data 5003 bit4=1 G91	INK	INK	ABS
Machine data 5003 Bit5=1 G90	ABS	INK	ABS
Machine data 5003 bit5=1 G91	INK	INK	ABS

INK = Incremental
ABS = Absolute

Example:

```
N1    G01  X50  Y60  F500 LF
N2    G91  X50  Y40  LF
N3    G90  X60  G91  Y30  LF
N4    M30  LF
```



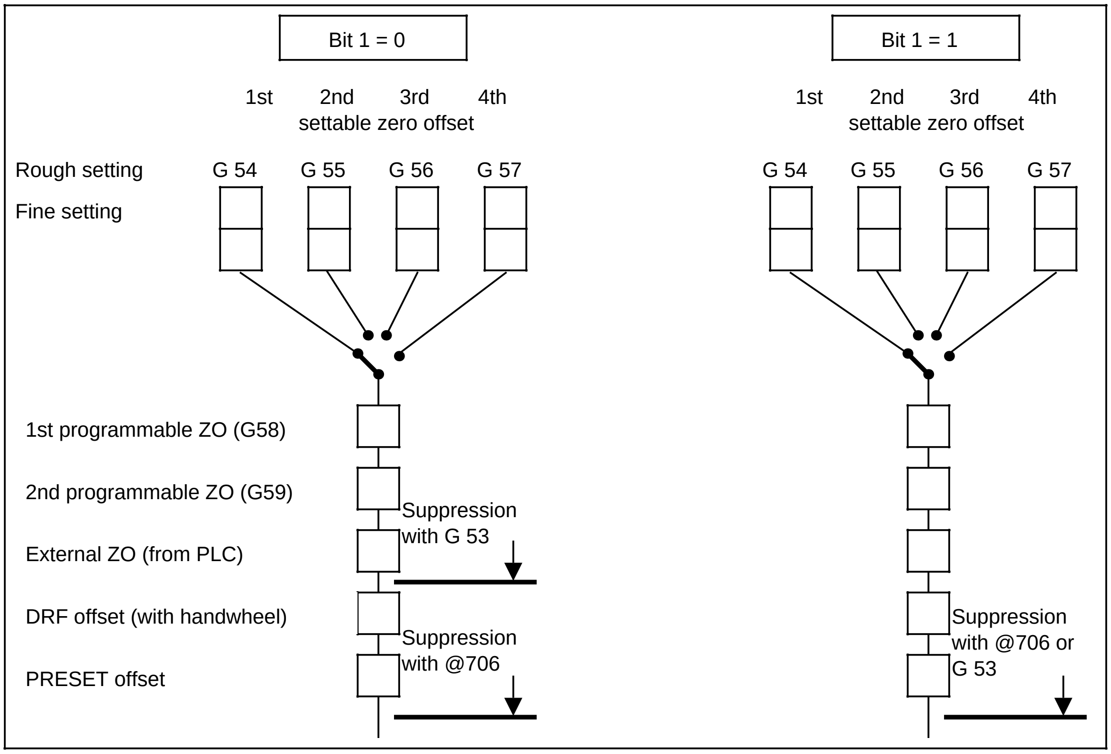
Bit 3: TO can be expanded by an additional two parameters (P8, P9) (see also Programming Guide).

Bit 2: **Bit 2 = 0** Subroutine end (M17) is issued to the PLC as an M function.

Bit 2 = 1 Subroutine end active only internally in NC (fast execution with subroutine return).

Bit 1: **Bit 1 = 0** With G53 all zero offsets (G54-G59 + ext. ZO) are cancelled.

Bit 1 = 1 With G53 and @706 all zero offsets (G54-G59 + ext. ZO), DRF and PRESET are cancelled. Tool offset (TO) is not cancelled.



Bit 0: Scan tool length compensation and zero offset even if axes are not programmed (only SINUMERIK 880 GA2 T):

If the change in tool length compensation (e.g. cancelled by means of D0) or change in a ZO yields a traversing path in one of the first two axes, whereby the axis concerned 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 SINUMERIK 880 GA2T:

```
          :
N5 G18 G0 X0 Z0 LF   D
N10 G01 F200 F3 X10 LF                 D3 ...type 3 (turning tool)
N20 Z30 LF                             Length 1     30
                                       Length 2     20
          :
```

With the bit set, length compensations are already moved in block N10 to position Z20.

If the bit is not set, the Z axis is first moved in block N20 to position Z50 (Z30 + length compensation 1).

5008	NC MD						5008
Bit No.							
7	6	5	4	3	2	1	0
Path dim. from PLC without NC STOP	REPOS in JOG mode	INC and REF in JOG mode	Tool type 0 as for type 20	Tool offset value monitoring			

Active: Immediately

Bit 7: **Bit 7 = 0** Path dimension from PLC (via command channel) is only started in the modes AUTOM./MDA, if the preselected NC channel is in the NC STOP state. If the NC channel is only stopped with a read-in disable at the end of block, the path dimension is **not** started. For safety reasons we recommend that you do not to set bit 7.

Bit 7 = 1 Path dimension from PLC (via command channel) is started in modes AUTOM./MDA on NC STOP or with read-in disable at end of block. It might happen that path dimension selected from the PLC sets the read-in disable and the running part program block is prematurely aborted by NC STOP (tool failure, ”NC STOP” key, ...). Now the function path dimension would be started from the PLC immediately and not at the end of the block. For this reason we recommend that you do **not** set bit 7.

Bit 6: **Bit 6 = 0** In REPOS mode the path back to the contour is started by pressing the appropriate direction key for a short time. The axis movement can now be stopped only with feed hold or feed override 0%.

Bit 6 = 1 You can only reapproach the contour as long as you are holding the appropriate direction key pressed.

Bit 5: As for bit 6 but for modes INC1 ... 10 000 and reference point, approach.

Bit 4: This bit can be used to determine whether an explicit input of the type is required in the case of milling cutters (tool type 20).

Bit 4 = 1 If the value 0 or 20 is in tool parameter P1, this is defined as the milling cutter. Note NC MD 5007 bit 7.

Bit 3 **Bit 3= 0** Monitoring of tool offset values

Values entered in tool wear memories P5, P6 and P7 are checked by the input key and EDIT key for their maximum values. If the maximum value is exceeded, this is diplayed in the input line. The maximum values are set in machine data MD 208/MD 209. The maximum values are: +/-99999999 +/-999.99999.

Tool offset parmeters P0 and P1 are also monitored.

Bit 3= 1 No tool offset values are monitored.

5009	NC MD						5009
Bit No.							
7	6	5	4	3	2	1	0
Double slide user interface	Graphic tool editor	Erase graphics	Data enable simulation	Blank optimiza-tion			

Bit 7: The user interface in graphic simulation can be selected for a double slide machine (single spindle) (precondition: option "graphic simulation").

SINUMERIK 880 GA2 T: 0 . . . . . Single slide
1 . . . . . Double slide single spindle

SINUMERIK 880 GA2 M: 0 . . . . . Milling simulation
1 . . . . . not permissible

Caution!

A modification of this NC MD only becomes active after POWER ON.

Bit 6: The graphic tool editor can be selected with the softkey "tool" in the menu tree for graphic simulation (precondition: Option "graphic simulation").
Without the tool editor the tool data can only be entered via lists or the serial interface.

Bit 5: You can enable broken line graphics and erase graphics (for turning graphics and milling graphics) (precondition: Option "graphic simulation").

Bit 4: With this bit simultaneous simulation (simulation simultaneous to processing) is always enabled.

This therefore determines that write accesses to data to which the NC channels also have access are permitted.

Same data for simulation and NC channels can be:

- R parameters R900 to R999
- MIB parameters
- Settable zero offsets G54 to G57
- Scale centre
- Scale factor
- NC setting data bits 5000 to 5999
- NC machine data
- PLC machine data
- Tool offsets.

If access is enabled, the operator has to make sure that the simulation does not overwrite any data of the NC channels (NC and simulation channels must not access the same data).

Bit 3: Construction of the picture and cyclic updating of the picture is accelerated noticeably if special routines are activated for blanks. This bit is set as a standard.

5010	NC-MD						5010
Bit No.							
7	6	5	4	3	2	1	0
				5-axis tool length compensation			

Active: In next block

	Bit 3	Bit 2	Bit 1	Bit 0
Gimbal head	0	0	0	1
Twist & nod head	0	0	1	0
Nutating head	0	1	0	0
Inclinable head (parallel Y)	0	1	0	1
Inclinable head (parallel Z)	0	1	1	0

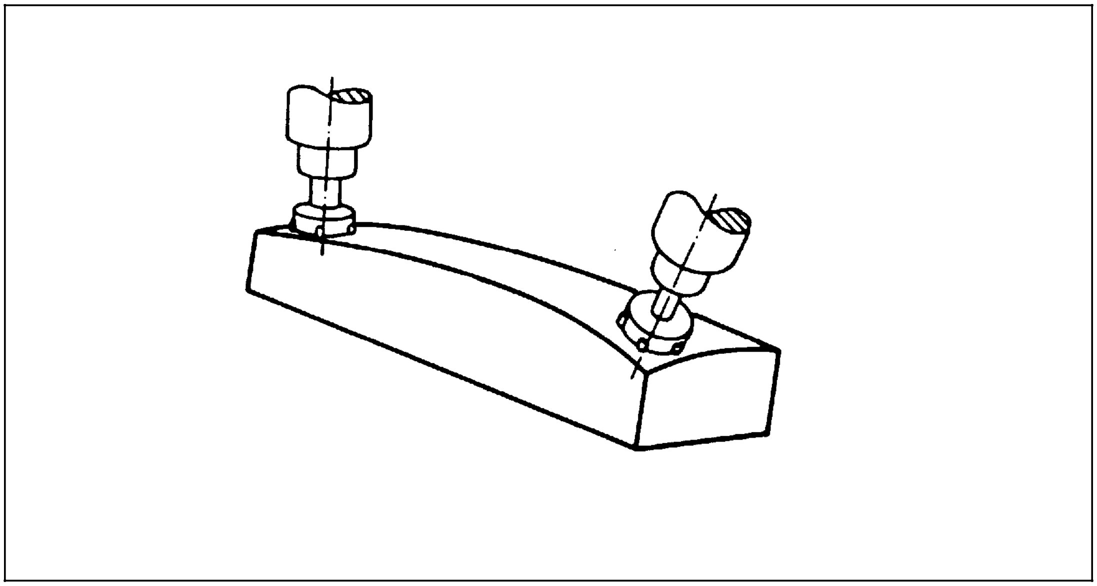
5-axis tool length compensation

5-axis tool length compensation is an Option.

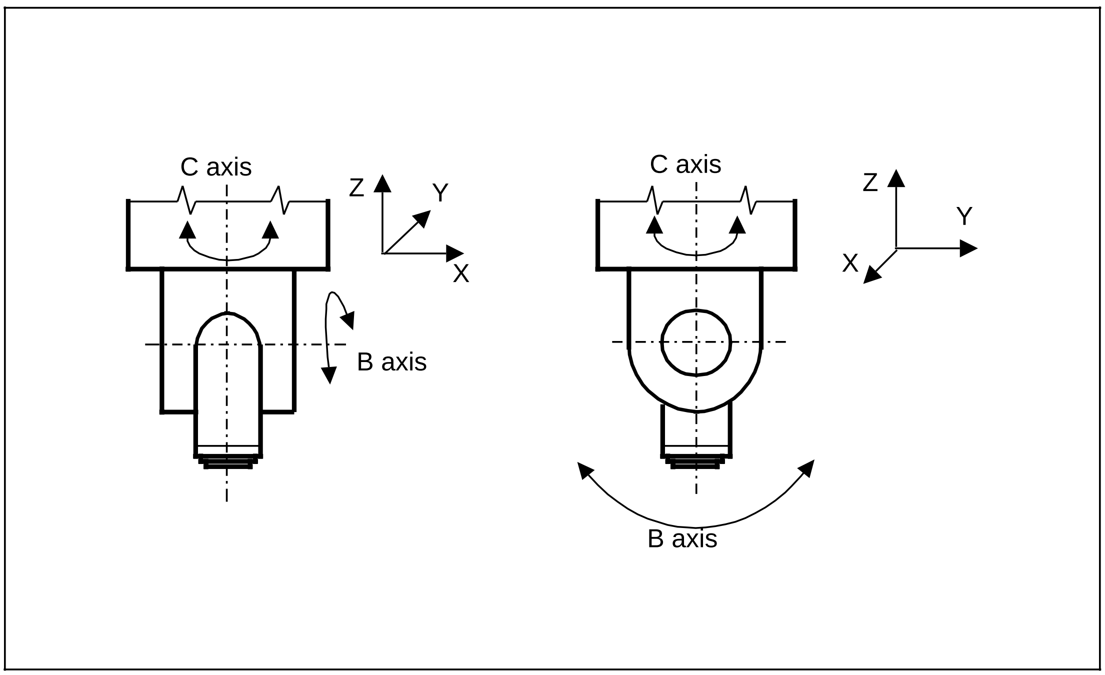
5-axis tool length compensation corrects the programmed block end points for short linear blocks (4 mm max.) as a function of the milling head type. In the event of tool failure, the milling machine can continue to operate at the failure point if a tool with the programmed radius but a different length is loaded. The differential length (50 mm max.) is entered as the tool compensation for tool type 40.

This assumes that the programs have been generated at the computer to incorporate the tool length and radius.

5-axis tool length compensation is called using MD 5010 and tool type selection 40. The tool length (differential length referred to old tool) is stored in TO parameter P2. In the case of milling machines with a nutating head, the fixed nutating angle must also be written in NC MD 20.

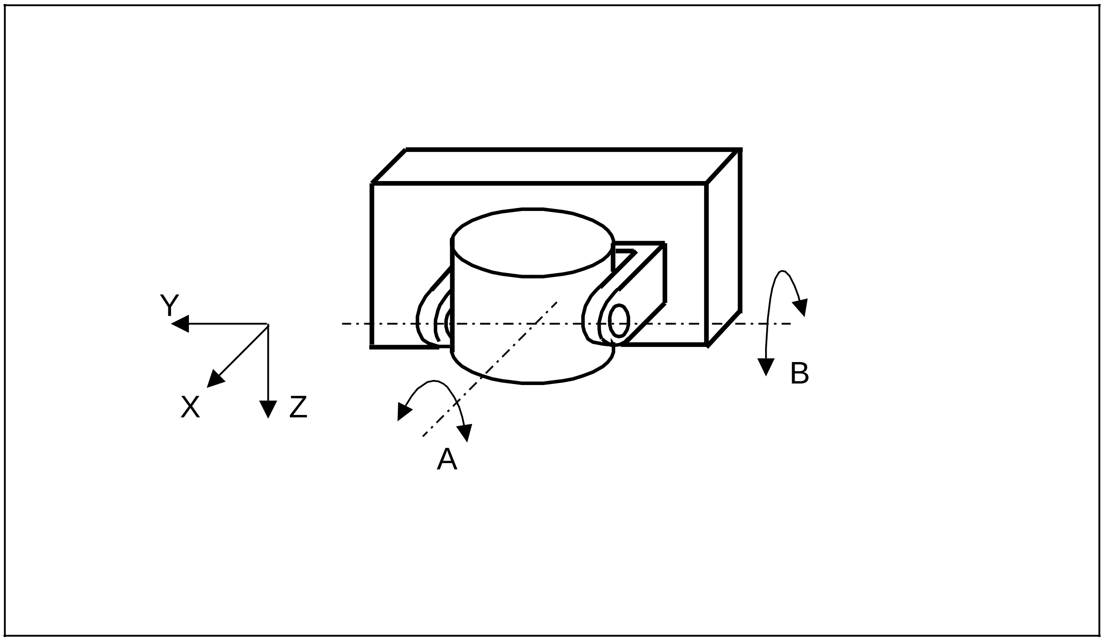


Principle of 5-axis tool length compensation (positioning of cutter in 5 axes)



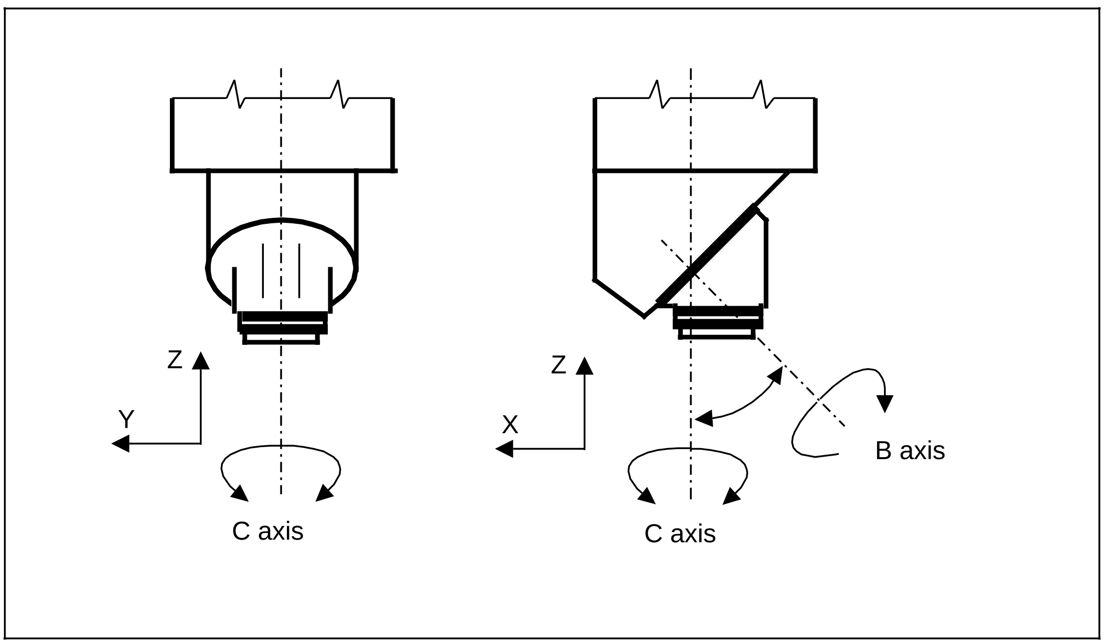
Gimbal head

The B axis is along the Y axis for C =0.
The tool is parallel to the Z axis if B =0.



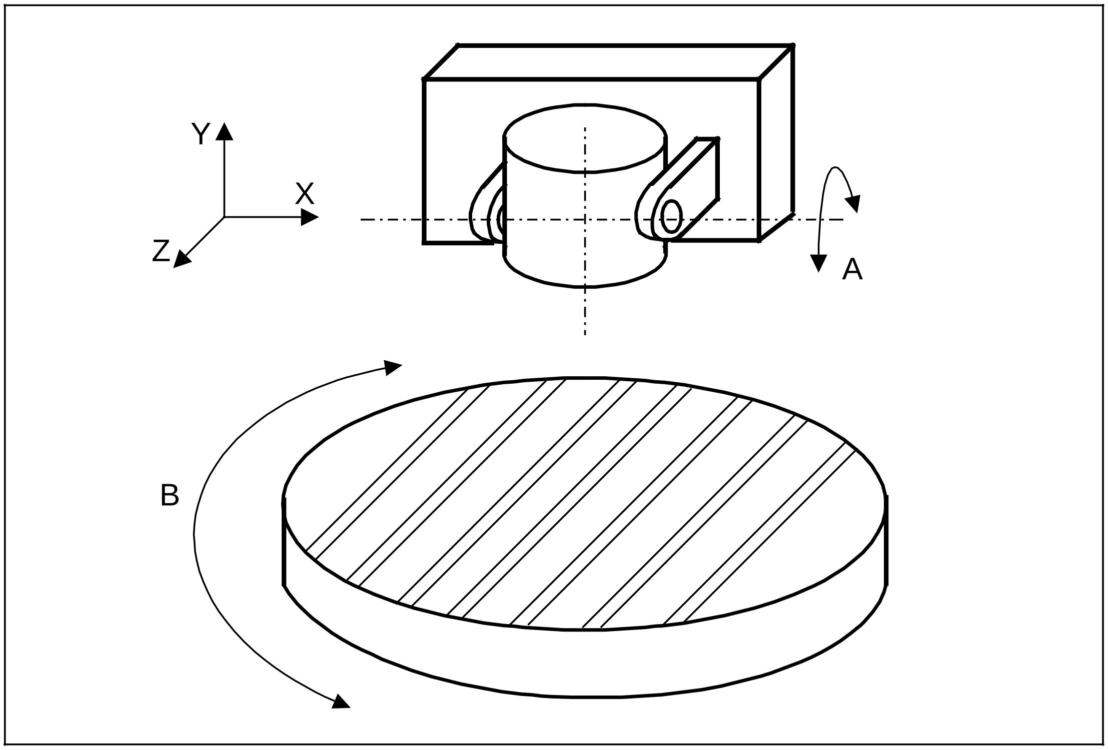
Twist and nod head

The tool is parallel to the Z axis if A = B = 0.
The A axis is always about the X axis.
The B axis is about the Y axis for A = 0.



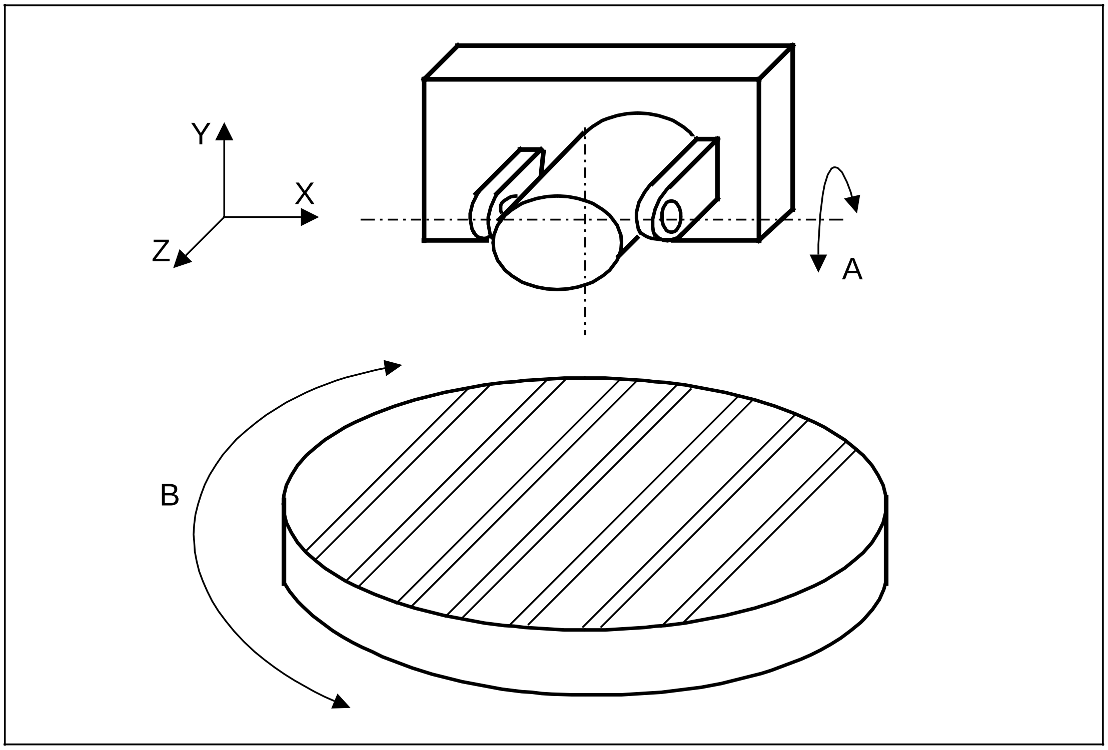
Nutating head

The tool is parallel to the Z axis for B = C = 0.
The B axis is in the X-Y plane for C = 0.



Inclinable head (cutter parallel to Y axis)

The rotary table (axis C) turns around the Z axis.
The swivel head (axis B) turns around the Y axis.
The swivel area around B is usually -45° to + 45°.
With A = 0 degrees the cutter is parallel to the Y axis.



Inclinable head (cutter parallel to Z axis)

The rotary table (axis C) turns around the Z axis.
The swivel head (axis B) turns around the Y axis.
The swivel area around B is normally -45° to + 45°.
If A = 0 degrees the cutter is parallel to the Z axis.

5011		NC MD Bit No.					5011
7	6	5	4	3	2	1	0
Read and write @ over diameter	Actual value display over diameter	Diameter prog. G91	Diameter prog. with G90 TO over diameter	Tool length for tool types 1-9 over diameter	Inc. handwheel DRF over diameter	Adj. ZO prog. ZO over diameter	

Active: In next block

All the bits in NC MD 5011 act on the axes defined as facing axes by NC MD 572* bit 1. If no facing axis was defined (NC MD 572* bit 1 = 0) all bits in NC MD 5011 must be set to zero.

Bit 7: All axis-specific @ for axes defined as a facing axis must be entered over the diameter and are written over the diameter. (CL 800)

Bit 6: The actual value and workpiece-related actual value are displayed over the diameter (see also NC SD 5001 bit 0).
The actual values in the axis-specific service displays are still shown as radii.

Bit 5: For G91 programming is performed over the diameter.

Bit 4: For G90 programming is performed over the diameter. Tool offset wear (P5) for tool types 1-9 is entered and displayed over the diameter.

Bit 3: The tool length (P2) for tool types 1-9 is entered and displayed over the diameter.

Bit 2: With INC handwheel and DRF handwheel, the input resolution is active over the diameter. DRF offset is displayed over the diameter.

Bit 1: Adjustable ZO and programmable ZO are entered, programmed and displayed over the diameter.
External ZO, PRESET offset, distance to go and REPOS are displayed over the diameter.

5012	NC MD Bit No.						5012
7	6	5	4	3	2	1	0
			Softkeys green inverse		MD modi-fication with @4.. disabled		

Bit 4: The softkey and the modes are always displayed INVERSE.

Bit 4=0: white inverse
Bit 4=1: green inverse

Active: After POWER ON

Bit 2: Writing from NC MD, PLC MD and cycle MD with @ commands @ 4... (CL 800 commands) can be disabled with bit 2.

Active: In next block

5013			NC MD Bit No.				5013
7	6	5	4	3	2	1	0
Circle radius programming			Feed not referred to contour		M and S address extension for spindle to PLC	Tapping without encoder (G63)	G63 without decel.

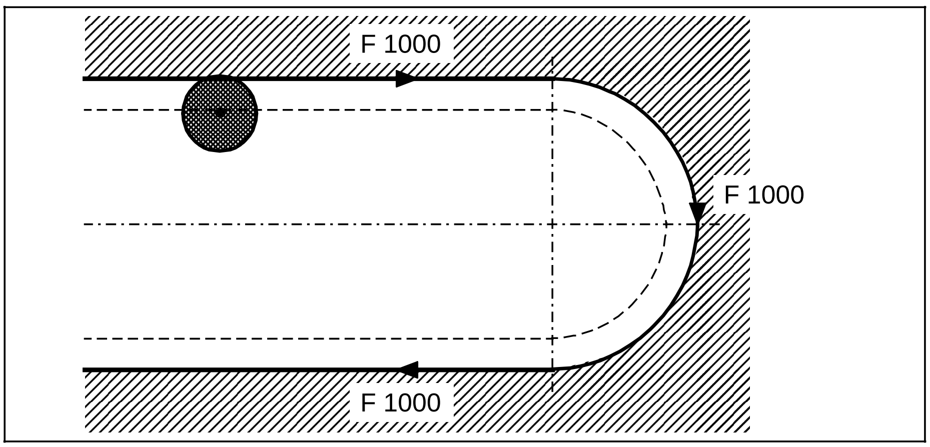
Active: In next block

Bit 7: With the bit set, a circle can be programmed by specifying the radius and/or angle.

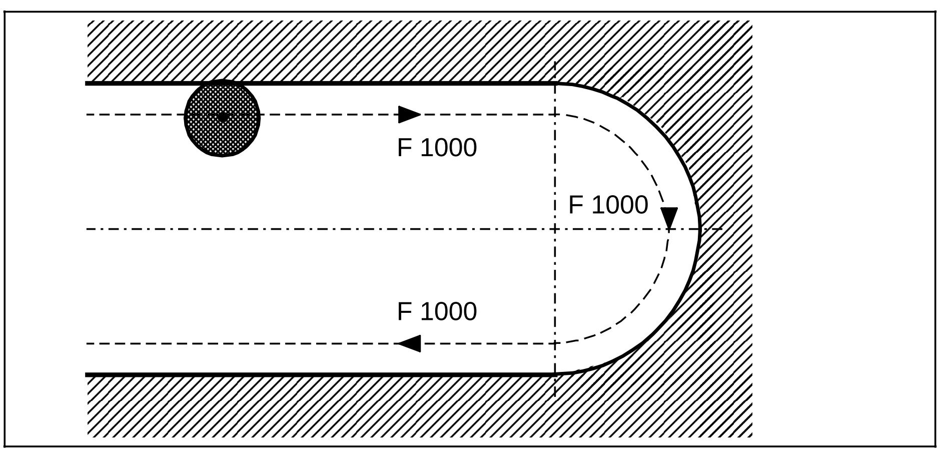
Bit 4: The programmed feedrate refers to the cutter centre path (tool nose radius centre path).

Example for SINUMERIK 880 GA2 M:

Bit 4 = 0 (feedrate at contour constant)



Bit 4 = 1 (feedrate of cutter constant)



Bit 2: This bit is used to select whether the programmed M and S address extension or the extension automatically generated by the controller is to be transferred to the interface to the PLC (this extension is always active in the control).

This bit must be set if there is more than one spindle in the control.

The leadscrew is automatically extended in the channel without MD 5013.2 being set.

Bit 1: For tapping with the Siemens L84 cycle (G84) using the SINUMERIK 880 M GA2, it is necessary to specify whether a encoder is available at the spindle.

Bit 1 = 0 The spindle has an encoder (512 or 1024 pulses). Therefore G33 (lead in mm/rev) is used in tapping cycle L84.

Bit 1 = 1 The spindle does not have an encoder or the available encoder is not to be used for cycle L84.. Therefore G63 (F in mm/min) is used in tapping cycle L84. The programmer has to specify the feedrate and spindle speed in such a way that the correct thread lead is obtained. Minor errors are eliminated by the compensating chuck.

Bit 0: At the block limits in G63 programming (tapping without encoder), it is possible to determine if the new spindle speed setpoint value (reversal) is output to the drive and a block change initiated.

Bit 0 = 0 The block change and the output of a new spindle speed setpoint value occur after exact positioning fine (drilling axis) and the acknowledgement of the M function (spindle reversal) via the PLC.

Bit 0 = 1 The block change and the output of a new spindle speed setpoint value occur when the interpolator (not the position controller with exact positioning fine) has reached the programmed drilling depth (i.e. sooner than the position controller with exact positioning fine). The acknowledgement of the M function (spindle reversal) via the PLC does not occur in this case.

5014	**NC MD**						**5014**
	Bit No.						
7	6	5	4	3	2	1	0
TNRC/ CRC		Cycles (ref. prep.)					

Active: After RESET

Bit 7: TNRC/CRC T: Activating of tool nose radius compensation

M: Activating of cutter radius compensation

Bit 5: Cycles: This bit activates reference preparation (software submodule) which is essential for cycle processing with stock removal cycle L95.

5015	NC MD						5015
Bit No.							
7	6	5	4	3	2	1	0
	UMS available			Speed optimized text output	Graphics		

Bit 6: The UMS (**u**ser **m**emory **s**ubmodule) with 128 K RAM (6FX1126-6BA), 128 K EPROM (6FX1128-4BD) and 256 K EPROM (6FX1128-4BC) configurations is inserted in the 6FX1126-7BA memory module in the COM area on the 1st slide-in unit. In the case of the RAM configuration, UMS data transfer is only possible in commissioning mode. The NC interface is to be set to line-driven data transfer, 2 stop bits without parity and 9600 baud. NC UMS output is initiated via WS 800. The Universal Interface Configuring Instructions contain information on cable connections.

If an attempt is made in commissioning mode to read in a UMS and if no USM RAM submodule is inserted or if a UMS EPROM submodule is inserted, Alarm 27 "Data input disabled RS232C (V.24)" and the dialog text "Data transfer error" are output.

Active: After POWER ON

Bit 3: Picture construction can be accelerated for graphic images if text is entered directly into text fields without first deleting the complete text field.

However, unforeseen mixed colours can arise if the background colour of the text field (even of parts of the text field) is not black.

Active: When new display is selected

Bit 2: The static graphics can be activated (not graphic simulationwith this bit). This bit must be set if the Siemens Standard UMS is to be used in its full functional scope.

Active: When new display is selected

5021-5023			NC MD Bit No.			5021-5023	
7	6	5	4	3	2	1	0
			FA overlay division-related	Suppression of acceleration limitation	Block change with fine synchronism	Sensitivity attenuation compensatory controller	Following error compensation ON

Caution:

These MDs have a different meaning with the sychronous mode function (see Function Manual "Extended spindle functions").

Active: Immediately

Bit 0: "Following error compensation ON"

1 signal: The process for following error compensation is active.
Following error compensation must always be active with ELG/sychronous spindle functionality. It can be activated in parallel with the feedforward control. The feedforward control must however first have been set.

0 signal: The process for following error compensation is not active.

Comment:

- The process for following error compensation is used only for actual value linked leading axes and also when changing over the following axes to "Automatically controlled correction".

Bit 1: "Sensitivity attenuation compensatory controller"

1 signal: The sensitivity for the compensatory controller response is attenuated. The P and D components of the compensatory controller respond only when speed deviations greater than 10 increments occur. This allows a smoother setting of the running characteristic for the following axis.

0 signal: The compensatory controller responds even to the smallest speed deviations and attempts to compensate for these by means of an additive set speed for the following axis.

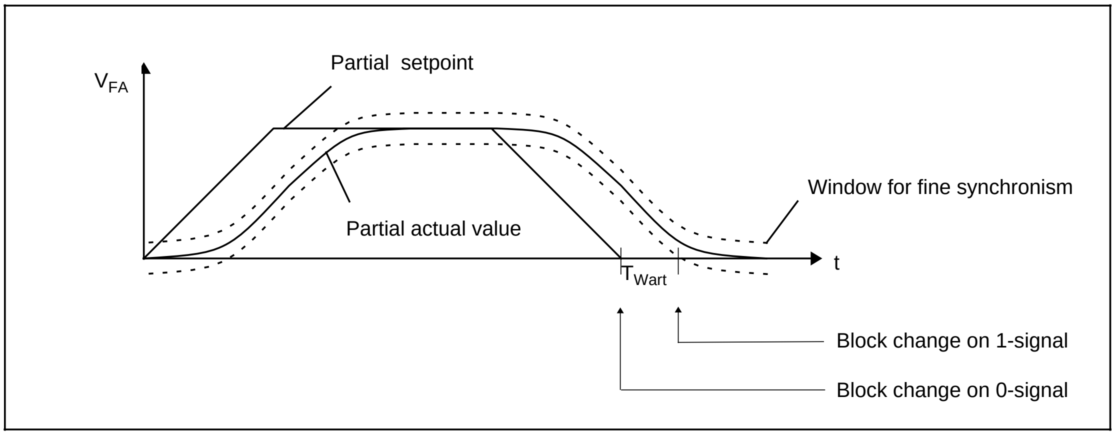
Note:

When loading the standard machine data, the machine data bit is preset with "1" signal.

Bit 2: If this bit is set, by G201 with no angle, a block change in the NC is initiated when the following spindle reaches "Fine synchronism" after runup.
(Note NC MD 952, 953, 972, 973, 992, 993 as regards the tolerance band).

If the bit is not set, the block change is initiated after a fixed delay (the settling time) when synchronous speed is reached. In the case of overlaid following axes a block change is initiated if:

- All partial setpoints of the block are output
- A delay of 4 x MD950 has elapsed
- Fine synchronism has been selected.



Bit 3: "Suppression of acceleration limitation"

1 signal: "Acceleration limitation is not active"
With LINK ACTIVE, the acceleration of the following axis is not limited by the control but output directly, as determined by the leading axes. If the value of MD 276* is exceeded, no alarm message is output.

0 signal: "Acceleration limitation active"
With LINK ACTIVE, the acceleration of the following axis is limited to the acceleration value set by machine data (MD 276*).

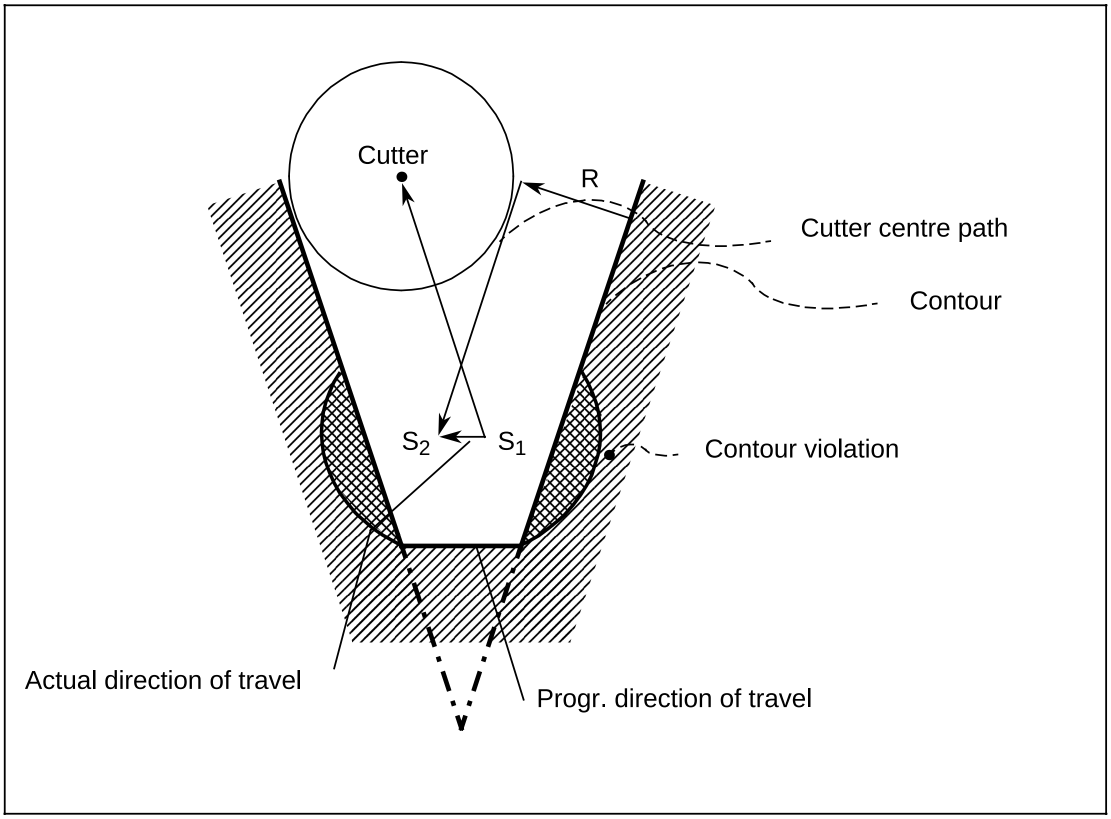
Bit 4: "FA overlay division-related"

1 signal: The overlaying movement of the following axis takes place in the indexing grid. On the other hand, the movements of the following axis caused by the leading axis are not used to determine the set indexing position. Linking and synchronization operate unchanged. The following axis must be defined as an indexing axis. The indexing grid is defined with MD 1104* and MD 1108*.

0 signal: The indexing grid is not taken in account for overlaying movements of the following axis.

5024	NC MD						5024
			Bit No.				
7	6	5	4	3	2	1	0
							Abort if contour violated Alarm 3021

Bit 0: If, during machining with active tool nose/cutter radius compensation (G41/42), a movement occurs in the opposite direction to the programmed direction of travel, alarm 3021 is set.



From the example it is clear that the contour (flanks of the groove) will be damaged.

Bit 0 = 0 With contour monitoring by the tool nose/cutter radius compensation (TNRC/CRC) machining is aborted. Alarm 3021 is given and can be cleared with the acknowledgement key.

Bit 0 = 1 With contour monitoring by the tool nose/cutter radius compensation (TNRC/CRC) machining is aborted. Alarm 3021 is given and can only be cleared with the reset key.

5030		NC MD Bit No.					5030
7	6	5	4	3	2	1	0
	Declaration of operator panel CPUs (OP CPUs)						
					BT 3	BT 2	BT 1

Active: After POWER ON

Bit 0: **Bit = 1** Number of operator panels (CPUs) is specified.

Bit = 0 Corresponding OP CPU is not available.

Caution:

Only declared CPUs are monitored in case of failure.

5031		NC MD Bit No.					5031
7	6	5	4	3	2	1	0
	Operator panel (OP CPU) configuration display						
					BT 3	BT 2	BT 1

With these bits, the control indicates how many operator panels are connected or how many operator panels it recognizes irrespective of NC MD 5030.

5032	NC MD						5032
7	6	5	4	3	2	1	0
	Declaration of COM CPU						COM CPU 1

Active: After POWER ON

With this data, the person installing specifies the number of COM CPUs available in the control.

Mandatory input on SINUMERIK 880 GA2: 0000 0001

5033	NC MD						5033
7	6	5	4	3	2	1	0
	COM CPU configuration display						COM CPU 1

With this data, the control indicates how many COM CPUs it recognizes.

5034			NC MD Bit No.				5034
7	6	5	4	3	2	1	0
	Declaration of NC CPUs						
						NC CPU 2	NC CPU 1

Active: After POWER ON

With this data, the person installing specifies the number of NC CPUs.

For SINUMERIK 880 GA2 max. two NC CPUs.

Caution:

Only declared CPUs are monitored in case of failure.

5035			NC MD Bit No.				5035
7	6	5	4	3	2	1	0
	NC CPUs configuration display						
						NC CPU 2	NC CPU 1

With this data, the control indicates how many NC CPUs it recognizes.

5036			NC MD Bit No.				5036
7	6	5	4	3	2	1	0
	Declaration of servo/processor submodules						
				Servo CPU 4	Servo CPU 3	Servo CPU 2	Servo CPU 1

Active: After POWER ON

With this data, the person installing specifies the number of servo/processor submodules.
For SINUMERIK 880 GA2 max. four servo/processor submodules.

5037	NC MD						5037
Bit No.							
7	6	5	4	3	2	1	0
	Servo/processor submodules configuration display						
				Servo CPU 4	Servo CPU 3	Servo CPU 2	Servo CPU 1

With this data, the control indicates how many servo/processor submodules it recognizes. The servo CPU houses only the axis position control / spindle control (also see NC MD 155, 160 to 163).

Caution:
Only declared CPUs are monitored in case of failure

5038	NC MD						5038
Bit No.							
7	6	5	4	3	2	1	0
	Declaration of PLC CPUs						
						PLC CPU 2	PLC CPU 1

Active: After POWER ON

With this data, the person installing specifies the number of PLC CPUs.

Caution!
With bit 1 = 0 (no PLC CPUs) the relevant PLC MD must be set to 0.

Note:
Up to two PLC CPUs (2x 135 WB) are possible or one external S5-155U.

5039	NC MD						5039
Bit No.							
7	6	5	4	3	2	1	0
	PLC CPUs configuration display						
						PLC CPU 2	PLC CPU 1

With this data, the control indicates how many PLC CPUs it recognizes.

Note:
Up to two PLC CPUs (2x 135 WB) are possible or one external S5-155U.

5040-5044	NC MD Bit No.						5040-5044
7	6	5	4	3	2	1	0
	User option bits for menu selection (WS 800)						

Active: After simulation start

See WS 800 description.

NC MD	Bit No.							
	7	6	5	4	3	2	1	0
5045		Simultaneous tool paths (broken line graphics)						
				Settable ZO	Progr. ZO	Length correction		
5046		Simulation actual value display (broken line graphics)						
				Settable ZO	Progr. ZO	Length correction	CRC	
5047		Simulation tool paths (animated graphics)						
				Settable ZO	Progr. ZO	Length correction	CRC	
5048		Simulation actual value display (animated graphics)						
				Settable ZO	Progr. ZO	Length correction	CRC	

NC MD	Bit No. 7	6	5	4	3	2	1	0
5045				0	1	0	0	0
5046				0	1	0	0	0
5047				0	1	1	1	0
5048				0	1	0	0	0

Active: After simulation start

By setting these bits you can determine whether on graphics simulation the actual value display and the display of the tool or the display of the tools is to be executed with or without calculation of the offsets (CRC, TNRC, tool length correction, programmable ZO, settable ZO). You must set separately for broken line graphics (NC MD 5045, 5046) and animated graphics (NC MD 5047, 5048).

Bit 4: The settable zero offsets G54 and G57 are included in the calculation if the corresponding enable is given in the simulation data image as well (see also Operator's Guide).
The settable coordinate system rotation is simulated independently of the NC MD bit.
The external zero offset, the PRESET offset and the DRF offset are generally **not** calculated.

Bit 3: As bit 4 but for programmable zero offsets.
The programmable coordinate system rotation is always simulated independently of the NC MD bit.

Bit 2: The tool length corrections (tool parameters P2 and P3) of the current D No. are included in the calculation. The tool wear (tool parameters P5 to P7) and the tool basic dimension (tool parameters P8 and P9) are generally **not** included in the calculation.

Bit 1: With cutters (tool type 20) the programmed cutter radius compensation (CRC) G41 or G42 is taken into account. If the bit is **not** set, G41 or G42 are displayed like G40.
With NC MD 5045 this bit does not exist.
The tool nose radius compensation (TNRC) is generally **not** taken into account (also see Operator's Guide).

5049	NC MD						5049
Bit No.							
7	6	5	4	3	2	1	0
						Service control	

Bit 1: With the Teleservice functionality, the two controls that are involved must be assigned different parameters, either as a customer control or as a service control. This bit is responsible for the differentiation.

Bit 1 = 0 : Customer control

Bit 1 = 1 : Service control

5051	NC MD						5051
Bit No.							
7	6	5	4	3	2	1	0
	Keyswitch active for						Cycles SD ELG IS

With cycle setting data, by setting the corresponding NC MD bits input or modification via the NC operator panel can be disabled with the keyswitch.
With the electronic gearbox activated, the interface signals (DB 29 "Signals to NC") are thus disabled. Operation from the PLC is then no longer possibile.

NC MD	Bit No. 7	6	5	4	3	2	1	0
5060		Channel number of the transformation						
5061		G function for transformation selection						
5062		Axis name 1st fictitious axis						
5063		Axis name 2nd fictitious axis						
5064		Axis name 3rd fictitious axis						
5065		Axis name 1st real axis						
5066		Axis name 2nd real axis						
5067		Axis name 3rd real axis						
5068		Axis name 4th real axis						
5069		Axis name 5th real axis						

Active: After warm restart

NC MD 5060 to 5069 . . . . . . 1st transformation data block
NC MD 5070 to 5079 . . . . . . 2nd transformation data block
. .
. .
. .
. .
. .
. .
NC MD 5130 to 5139 . . . . . . 8th transformation data block

For an exact description of the coordinate transformation TRANSMIT see Section 11.

Standard MD for a transformation record:

NC MD 5060	0000 0000
NC MD 5061	0000 0000
NC MD 5062	1111 1111
NC MD 5063	1111 1111
NC MD 5064	1111 1111
NC MD 5065	1111 1111
NC MD 5066	1111 1111
NC MD 5067	1111 1111
NC MD 5068	1111 1111
NC MD 5069	1111 1111

5141-5146		NC MD Bit No.				5141-5146	
7	6	5	4	3	2	1	0
	Ethernet address						

Active: After POWER ON

The Ethernet address is the address of the bus interface through which it can be addressed.

Caution!

The Ethernet address in the MD is only valid if no valid database is available on CP 231 (CP in stop).

NC MD	Ethernet address		
	(Binary)	Byte	Decimal
5141	0 0 0 0 1 0 0 0	Byte 1	08
5142	0 0 0 0 0 0 0 0	Byte 2	00
5143	0 0 0 0 0 1 1 0	Byte 3	06
5144	0 0 0 0 0 0 0 1	Byte 4	01
5145	selectable	Byte 5	selectable
5146	selectable	Byte 6	selectable

The last four places of the Ethernet address must be issued for each bus interface.

5147	NC MD Bit No.						5147
7	6	5	4	3	2	1	0
				Changes when subroutines are read-in via file transfer	Do not erase erase-protected programs	Acknow-ledge message immediately	Positive acknow-ledgement to end message

Active: After POWER ON

Bit 3=1: The program which is to be read in is not read in meaning that the EPROM cycle is still free for processing. Cancel alarm 3240 "SPF not read-in" is output. The program number in question is output in the block number of the alarm. Subroutines continue to be read in via RS232C/file transfer.

Bit 3=0 The program to be read in is read in and is ready for processing. Cancel alarm 3239, "EPROM cycle overwritten by SPF" is output. The program number in question is output in the block number of the alarm.

Bit 2: Programs being processed are not erased when an erase command arrives via transfer file. If the MD is **not** set the programs are not erased until they have been enabled for erasing (at end of processing or on reset). No further messages are processed for file transfer.

Bit 1: On receipt of a message from the external partner (RNaaa__) a request for transmission of further data is already sent to the external partner (TNaaa__) before the data is entered in the memory. In this way the external partner can already be sending further data to the NC while the NC is still entering data into memory. If an error occurs while the data is being entered into memory, the message with the error is acknowledged as previously. The next message, which has already been requested and might be waiting in the dual-port RAM, is acknowledged with the error message "Function cannot be processed".

Bit 0: On receipt of an error-free end message a positive acknowledgement (RNaaa _E_) is sent from the external partner (REaaa__). The NC also expects a positive acknowledgement (REaaa__) from the external partner if the NC has sent an end telegram to the external partner. Changes occur when subroutines are read in via RS232C/file transfer.

Location receiver

5148-5151	NC MD Bit No.						5148-5151
7	6	5	4	3	2	1	0
See below							

MD No.:	7	6	5	4	3	2	1	0		ASCII
5148	0	1	0	0	0	1	1	0	≙	F
5149	0	1	0	0	1	1	0	0	≙	L
5150	0	1	0	1	0	0	1	0	≙	R
5151	0	1	0	1	1	1	1	1	≙	Underline

These machine data are only valid for "Execution from external" via the computer link interface.

The "logical partner receiver" is entered in the 4 bytes to which the request message is sent when the "Execution from external" function is started. The values are entered as ASCII values (see above). The standard setting is "FLR _" (see above).

Location receiver

5152	NC MD Bit No.						5152
7	6	5	4	3	2	1	0
0	0	0	1	0	0	0	1

This machine data is only valid for "Execution from external" via the computer link interface.

The "location receiver" is entered in this byte which indicates the interface module via which the "logical partner receiver" can be reached.

The following applies:	11 H (0001 0001)	1st interface module
	12 H (0001 0010)	2nd interface module
The standard setting is:	11 H (0001 0001) see above.	
Additional machine data:	(MD 30, MD 130*, MD 5148-5151)	

5156-5182			NC MD Bit No.				5156-5182
7	6	5	4	3	2	1	0
	Coupled axis combination						

Active: After warm restart

The definition of this coupled axis combination defines which coupled axis pairings form a coupled axis combination and how the coupled motion is performed (coupled motion in the same direction or opposite directions).

The 9 possible coupled axis combinations are programmed with G151 to G159. Each coupled axis combination is defined with 3 NC MDs corresponding to the 12 coupled axis pairings (12 times 2 bits).

	Programming	NC MD
1st coupled axis combination	G151	5156 to 5158
2nd coupled axis combination	G152	5159 to 5161
3rd coupled axis combination	G153	5162 to 5164
4th coupled axis combination	G154	5165 to 5167
5th coupled axis combination	G155	5168 to 5170
6th coupled axis combination	G156	5171 to 5173
7th coupled axis combination	G157	5174 to 5176
8th coupled axis combination	G158	5177 to 5179
9th coupled axis combination	G159	5180 to 5182

Two bits are assigned to every coupled axis pairing in every coupled axis combination.

Leading axis	898	896	894	892	890	888	886	884	882	880	878	876	**NC MD**
Coupl. mot. axis	899	897	895	893	891	889	887	885	883	881	879	877	**NC MD**
Definition of the coupled axis pairing	Bit				Bit				Bit				
	7 and 6	5 and 4	3 and 2	1 and 0	7 and 6	5 and 4	3 and 2	1 and 0	7 and 6	5 and 4	3 and 2	1 and 0	
Programming function													
G 151	5158				5157				5156				
G 152	5161				5160				5159				
G 153	5164				5163				5162				
G 154	5167				5166				5165				**NC MD**
G 155	5170				5169				5168				
G 156	5173				5172				5171				
G 157	5176				5175				5174				
G 158	5179				5178				5177				
G 159	5182				5181				5180				

With these 2 bits you determine how coupled motion is to be performed.

00 ... no coupled motion

10 ... coupled motion in opposite directions $\rightleftarrows$

11 ... coupled motion in the same direction $\rightrightarrows$

01 ... not defined (Alarm 85)

The definition of the coupled motion combination can be changed on a warm restart without POWER ON (for warm restart see also Section 11.6).

5183	NC MD						5183
	Bit No.						
7	6	5	4	3	2	1	0
2nd/3rd operator panel; transfer customer keys							
8th customer key	7th customer key	6th customer key	5th customer key	4th customer key	3rd customer key	2nd customer key	1st customer key

5184	NC MD						5184
	Bit No.						
7	6	5	4	3	2	1	0
2nd/3rd operator panel; transfer customer keys							
		14th customer key	13th customer key	12th customer key	11th customer key	10th customer key	9th customer key

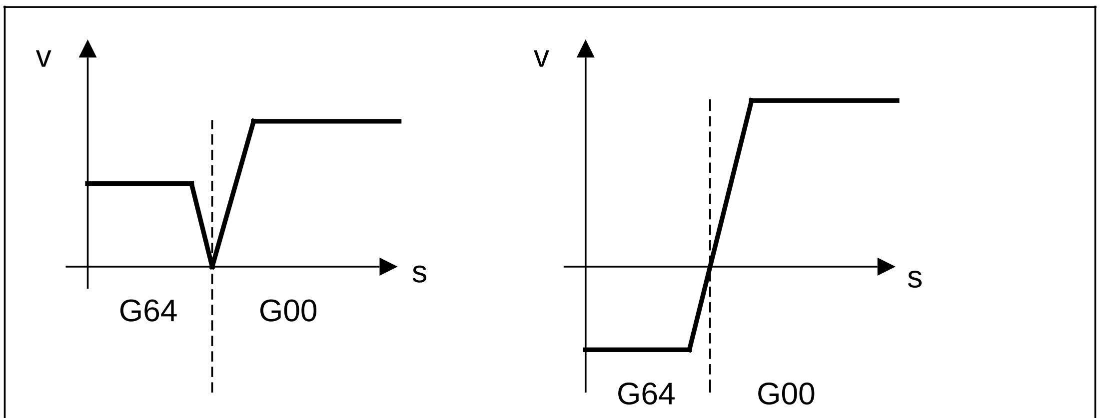
If these MD bits are set, the customer key functions of the 1st operator panel are automatically transferred by the PLC operator system to the customer keys of the 2nd/3rd operator panel when switching to the 2nd/3rd operator panel. This means that the selected keys are operational on all operator panels at the same time.

5185	NC MD Bit No.	5185

7	6	5	4	3	2	1	0
							Change from G64 to G00

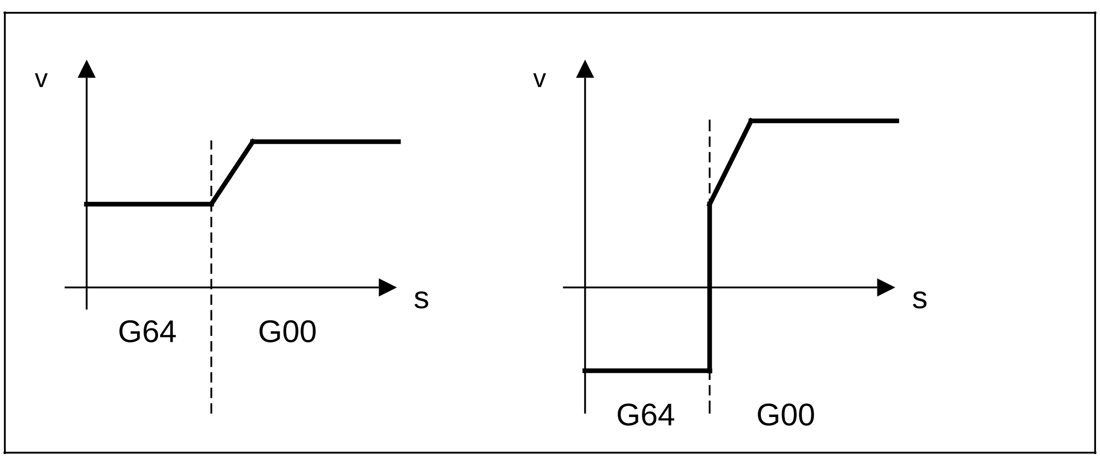
Active: In next block

Bit 0=1: If a path is traversed in one NC block with G64 and in the next NC block with G00, the NC automatically generates a speed reduction with exact stop "coarse" at the end of the first block.



Control generates exact stop "coarse" **Control generates exact stop "coarse"**

Bit 0 = 0:



520*	NC MD Bit No.						520*
7	6	5	4	3	2	1	0
Spindle override active with thread cutting	No M19 abort on RESET	M19 with axis movement		Speed in 0.1 rev/min.	Encoder available	Actual value sign change	

Bit 7: If this bit is **set**, speed override is also active for thread-cutting (G33, G37, G34, G35).

The change in spindle speed will cause a change in the following error which could lead to a fault in the thread.

Bit 6: If this bit is **set**, the positioning mode is not aborted by reset and M02/M30. The position is retained until IS: ACKNOWLEDGE M19 is set.

Bit 5: If this bit is **not set**, the next block in the part program will not be executed until the positioning mode has been aborted or concluded (with IS:ACKNOWLEDGE M19).

If this bit is **set**, the next block (with axis motion, if applicable, even without spindle controller enable) will also come up for execution even when the spindle is still being positioned. If there is an M03/M04 in one of the next blocks, it will abort the positioning mode.

This bit is generally used when deselecting the positioning mode. More details are given in "Special changeover conditions".

Active: Immediately

Bit 3: If this bit is not **set**, all speed data (for inputs, in machine data, etc.) are evaluated in the units rev/min.

If this bit is **set**, all speed data are evaluated in the units 0.1 rev/min.

Active: After POWER ON

Bit 2: The following spindle functions assume that a pulse encoder is mounted on the spindle:

• Thread-cutting	G33/G37/G34/G35
• Revolutional feedrate	G95
• Constant cutting rate	G96
• Accept spindle speed	G97
• Synchronous mode	G201
• Positioning mode	M19

This bit must be set if there is a pulse encoder mounted on the spindle.

Active: After POWER ON

Bit 1: If the value of this bit is changed, the sign will be inverted when evaluating the pulses coming from the measuring system.

Active: When all channels of the mode group are in STOP state

521*	NC-MD Bit No.						521*
7	6	5	4	3	2	1	0
Spindle available	No spindle speed change on reset	New S value after PLC acknow-ledgement	C axis operation cancelled after M02/M30	No measuring-circuit monitoring		Sign inversion, setpoint	

Bit 7: With SINUMERIK 880 GA2 there can be up to six spindles. The spindles must be defined in a continuous ascending sequence.

Any changes to this bit only become operative after a hardware reset.

Bit 6: If this bit is **set**, any running spindle functions are not aborted by RESET, mode change or M02/M30.

The spindle functions can then be aborted with IS:SPINDLE RESET provided, however, that the channel to which the spindle is assigned is in reset state.

For the positioning mode, MD 520*, bit 6 must also be taken into account.

All alarms which cause IS:MODE GROUP READY to be cancelled will stop the spindle in any case.

NC-MD 521* Bit 6 auf 1 (kein Stillsetzen der Spindel mit M30 und RESET)

Mode	RESET key	PLC oscillation/ basic	Spindle/ RESET DW K + 1.5	Program in channel	Spindle behaviour
JOG	–	Selected	Bit = 0	RESET	Overwritten with oscillation/basic speed
JOG	–	Deselected	Bit = 0	RESET	Previous speed effective
JOG	–	–	Bit = 1	RESET	Spindle at zero speed
JOG	–	Selected	Bit = 1	RESET	Spindle at zero speed
JOG	–	Deselected	Bit = 1	RESET	Spindle at zero speed
MDA/AUTO	–	–	Bit = 0	STOP	Selected speed effective
MDA/AUTO	–	–	Bit = 1	STOP	Selected speed effective
MDA/AUTO	Actuated	–	Bit = 1	RESET	Spindle at zero speed
MDA/AUTO	–	Selected	Bit = 0	STOP	Overwritten with oscillation/basic speed
MDA/AUTO	–	Deselected	Bit = 0	STOP	Previous speed effective
MDA/AUTO	–	Selected	Bit = 1	STOP	Overwritten with oscillation/basic speed
MDA/AUTO	–	Deselected	Bit = 1	STOP	Previous speed effective
MDA/AUTO	Actuated	Selected	Bit = 1	RESET	Spindle at zero speed
MDA/AUTO	Actuated	Deselected	Bit = 1	RESET	Spindle at zero speed
MDA/AUTO	Actuated	Selected	Bit = 0	RESET	Rotates at oscillation/basic speed
MDA/AUTO	Actuated	Deselected	Bit = 0	RESET	Spindle at zero speed

After spindle reset and spindle zero speed, M02/04 and S xxx have to be reselected.

If requested spindle positioning is aborted by the PLC with spindle reset, spindle positioning must be initiated again by the PLC.

Bit 5: If this bit is **set**, a newly-programmed spindle speed which requires a change of gear ratio will only be accepted when the gear ratio change has been acknowledged by the PLC. This enables any unwanted increase in speed in the old gear ratio to be avoided.

Bit 4: If the bit is **set**, C axis operation is cancelled after M02/M30 and after Reset (G 205 is generated).

Active: Immediately

Bit 3: If this bit is **set**, no measuring-circuit monitoring is performed.

Examples of such monitoring are:

- checking for open-circuit in encoder circuits
- checking for contamination, if the encoder possesses a signal for contamination, or if the amplitude can be monitored (e.g. signal generators).

Any changes to this bit only become operative after power-on.

Bit 1: If this bit is **not set**, a positive setpoint voltage will be output with M03 and a negative value with M04.

If this bit is **set**, a negative setpoint voltage will be output with M03 and a negative value with M04.

This allows the direction of spindle rotation to be matched to the programmed direction. With IS:INVERT M03/M04 the direction of rotation can be reversed by the PLC.

Active: When all channels of the mode group are in STOP state

524*	NC MD Bit No.						524*
7	6	5	4	3	2	1	0
				Position control resolution for spindle			

Active: After POWER ON

Bits 0 to 3: Position control resolution for spindle.

The position control resolution for the spindle set with MD 524* is valid for weighting one increment of the partial setpoint of the spindle in the positioning mode (C axis operation).

The selected position control resolution is matched to the rotary encoder with the aid of machine data 455* and 456*.

Refer to the table for the position control resolution of rotary axes for the coding of the possible position control resolutions (see MD 1800* bits 0 to 3).

If a C axis is assigned to the spindle, the spindle and C axis must have the same position control resolution.

540*	NC MD						540*
	Bit No.						
7	6	5	4	3	2	1	0
No transfor-mation deselection on RESET	F value in m/min	0*)	0*)			G functions to PLC	Aux. functions to PLC

Bit 7: On a key reset, mode change and M2/M30, the transformation (TRANSMIT) is not deselected.

Active: After RESET

Note:

In reference point approach mode the transformation is always deselected.

Bit 6: Initiates input of F value in m/min.
This machine data is active in all metric input systems. The maximum axial speeds stated in the machine data remain unaffected by the machine data (input units still "units").

Active: In next block

Input F10 speed 10 m/min.

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. This MD has no effect with ”inch” input resolution; i.e. the feedrate is always stated in inch/min.

Bit 1: **Bit 1=0** Inhibits output of G functions to PLC

Active: Immediately

Bit 0: **Bit 0=0** Inhibits output of auxiliary functions to PLC
Auxiliary functions are: M, S, T, H, D.
This is useful with calculation channels.

Active: Immediately

See NC MD 544* bit 0 for the output of the programmed F value to the PLC.

*) *This data has to be defined into value "0".*

542*	NC MD						542*
	Bit No.						
7	6	5	4	3	2	1	0
				Fast T function output	Fast S function output	Fast F function output	Fast D function output

With standard auxiliary function transfer, the NC holds the next part program block until the PLC has acknowledged the auxiliary function (after one PLC cycle at the soonest).

The function "Fast auxiliary function output to PLC" is provided to bypass this waiting time by means of an immediate acknowledgement.

Bits 0 to 3 are used to specify whether T, S, F or D functions are to be output fast (that is without a "reliable" acknowledgement from the PLC).

The fast transfer of M and H functions can be made block specific by programming a negative value (e.g. M = –7).

A block can contain several auxiliary functions. However, the fast output of auxiliary functions for a block is active only if all the auxiliary functions in the block are declared fast.

Bit 0 = 1: Fast output of all D functions

Bit 1 = 1: Fast output of all F functions

Bit 2 = 1: Fast output of all S functions

Bit 3 = 1: Fast output of all T functions

Notes:

- The auxiliary functions are not output fast in blocks with program coordination calls.
- Auxiliary functions that accumulate during block search are not output fast.
- Graphic simulation cannot handle fast auxiliary function output.
- If "fast auxiliary functions" intended to trigger the read-in disable are programmed in a block, the user is unable to interrupt part program processing with read-in disable **in the same block** (DB 10-25 DW 7-10).
- With H and F functions, the declaration can also be made with R parameters (e.g. M = 14; R14 can have a positive or negative value).
- The output of auxiliary functions has to be tied to the block (slow output) in part program blocks out of which G functions are to be read via FB 69.
- M02/M30 cannot be programmed as fast auxiliary functions.
- M00, M01 and M17 can be programmed as fast auxiliary functions.

544*	NC MD Bit No.						544*
7	6	5	4	3	2	1	0
Output of auxiliary functions in BCD							No F value output to PLC
F	H	D	T	S	M		

Active: In next block

Bit 3-7: **Bit 3-7 = 1** The auxiliary function is output to the PLC in BCD code.

Bit 3-7 = 0 The auxiliary function is output to the PLC as a fixed point number.

Bit 2: M function must be set to "0".
(internal M decoding)

Bit 1: Set to "0".

Bit 0: F value output to the PLC is cancelled via the machine data. Consequently, the block change times can be improved in the case of travel blocks with an F value since acknowledgement by the PLC does not have to be awaited (e.g. for 5D machining).

546*	NC MD						546*
Bit No.							
7	6	5	4	3	2	1	0
Auxiliary functions not collected during block search but output immediately							No aux. funct. output with block search
F	H	D	T	S	M		

Active: Immediately

Bit 7, 6, 5, 4, 3, 2:

The type of auxiliary functions specified is **not** collected with block search but is output immediately (may result in several switching functions being triggered in rapid succession via the PLC).

Bit 0: With the bit set, all auxiliary functions (M, S, T, D, H and F) are skipped and forgotten with block search (no output to the PLC and no internal response).

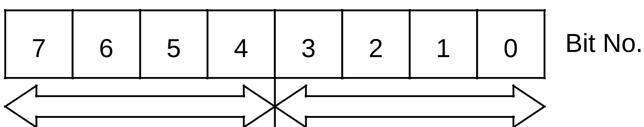
Example:

	Bit pattern for bits ...							**Significance**
	7	**6**	**5**	**4**	**3**	**2**	**0**	
Example a		irrelevant					1	No auxiliary function output
Example b	0	0	0	0	0	0	0	Collect all auxiliary functions and output last function after NC-Start
Example c	1	1	1	1	1	1	0	Output all auxiliary functions during block search
Example d	0	0	0	0	0	1	0	Output M functions during block search; collect H, D, T, S and F functions and output last auxiliary function after NC start.

All the collected auxiliary functions are activated at the end of block search with NC START and output to the PLC. Collecting ensures that only the last H, D, T, S and M functions and the last F value are active.

NC MD	Bit No. 7	6	5	4	3	2	1	0
548*	Name of horizontal axis (abscissa) (same code as for axis definition)							
550*	Name of perpendicular axis (ordinate) (same code as for axis definition)							
552*	Name of vertical axis (applicate) (same code as for axis definition)							

Active: In next bock



Extended address (address No.)

Key:

blank	=	0000
1	=	0001
2	=	0010
3	=	0011
4	=	0100
5	=	0101
6	=	0110
7	=	0111
8	=	1000
9	=	1001
10	=	1010
11	=	1011
12	=	1100
13	=	1101
14	=	1110
15	=	1111

Address name

Key:

X	=	0000
Y	=	0001
Z	=	0010
A	=	0011
B	=	0100
C	=	0101
U	=	0110
V	=	0111
W	=	1000
Q	=	1001
E	=	1010

Permissible names for axes, angles, chamfers and radius

A *Unassigned address*
B *Unassigned address*
C *Unassigned address*
D *Tool offset number*
E *Unassigned address*
F Feed
G G function
H H function
I Interpolation parameter
J Interpolation parameter
K Interpolation parameter
L Subroutine
M M function
N Subblock
O Danger of confusion with 0 (zero)
P Subroutine number of passes
Q *Unassigned address*
R Calculation parameter
S Spindle speed, S function
T Tool
U *Unassigned address*
V *Unassigned address*
W *Unassigned address*
X *Unassigned address*
Y *Unassigned address*
Z *Unassigned address*

The reset plane in the SINUMERIK 880 GA2 can be defined in NC MD 110*. After powering up the NC, the axes in which radius compensation or length compensation is to be calculated are specified as a function of NC MD 548*, 550* and 552*. NC MD 548* and 550* define the axes to be affected by radius compensation in the reset position. NC MD 552* defines the axis in which length compensation 1 (tool parameter P2) is to be active (cutters only). Calculation of a second length compensation (tool parameter P3) depends on the sequence of the programmed axes and type of tool.

Example: NC MD 548* = 0000 0000 X axis
NC MD 550* = 0000 0001 Y axis
NC MD 552* = 0000 0010 Z axis

- Program

N10 G0 G41 D1 X0 Y0 Z0 (D1 = tool type 20 cutter)
Radius is calculated in X-Y
L1 is calculated in Z
L2 is not calculated

- Program

N10 G16 X Y Y Z (plane selection)
N20 G0 G41 D1 X ... Y ... Z ... (D1 = tool type 30 angle head cutter)

Radius is calculated in X-Y
L1 is calculated in Y
L2 is calculated in Z

- Program

N10 G16 Z X Z (plane selection)
N20 G0 D1 Z ..... (D1 = tool type 10 drill)

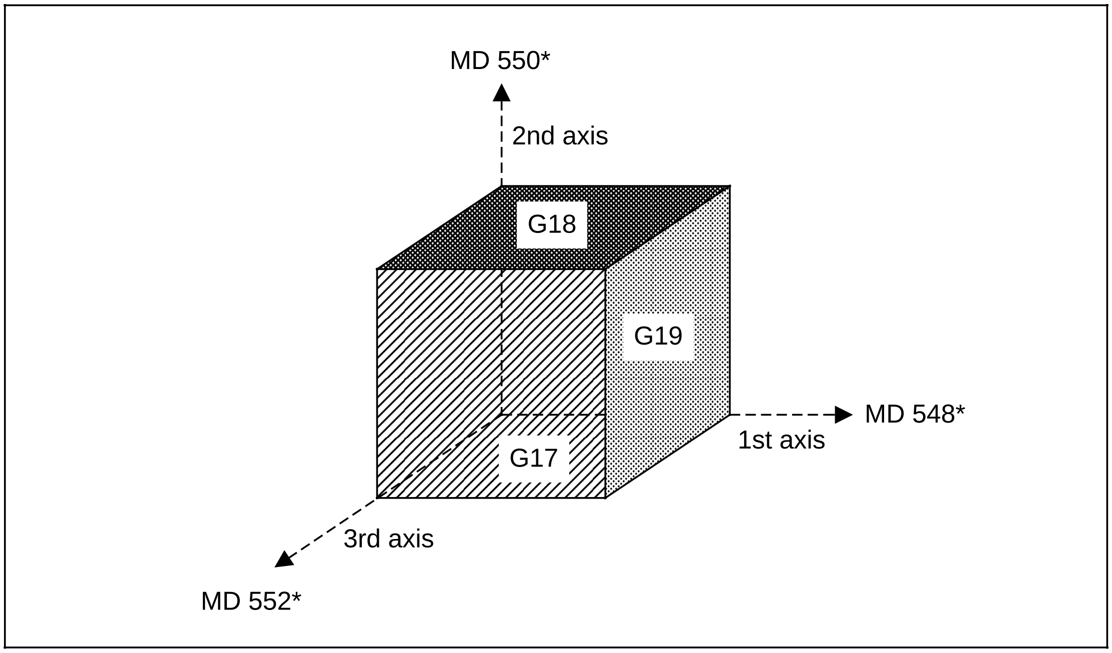
Radius is not calculated
L1 is calculated in Z

In a turning machine with X and Z axes, Z-X (G18) is to be selected as the reset plane after powering up. The NC MD should be set as follows:

NC MD 548* = 0000 0000
NC MD 550* = 0000 0010
NC MD 552* = 0000 0010

NC MD 548* and 550* specify plane Z-X. NC MD 550* specifies the axis in which L1 geometry is to be active if a cutter or drill is used.

With tool type 1 to 9 the following applies:	Length 1 (tool parameter P2) always refers to the 2nd axis name behind G16 <
	Length 2 (tool parameter P3) always refers to the 1xt axis name behind G16
	Length 1 (tool parameter P2) permanently refers to the facing axis.



PLANE	G17	G18	G19
Axis No.	1	3	2
	2	1	3
	3	2	1

554*	NC MD Bit No.						554*
7	6	5	4	3	2	1	0
Axis with constant cutting speed G96 (axis No. n-1)							

Active: In next bock

The **number** of the axis with which the constant cutting speed (G96) is to be reached must be specified (normally the plane axis).

The axis number must be specified as follows:

1st axis ... 0000 0000
2nd axis ... 0000 0001
3rd axis ... 0000 0010
4th axis ... 0000 0011
.
.
.
10th axis ... 0000 1001

558*	NC MD Bit No.						558*
7	6	5	4	3	2	1	0
Tapping without fine inter-polator						FIFO active	Supple-ment axes after block search

Active: In next block

Bit 0: **Bit 0 = 0** After block search only those axes programmed in the search block are traversed. It is possible to modify TO and ZO prior to NC START.

Bit 0 = 1 "3D interpolation "Option must be available. After block search, not only those axes programmed in the search block are traversed; up to 5 axes may be traversed if previously entered in the program.
When supplementing the axes, reference is mode only to the first 5 axes defined at the machine (NC MD 200*).
If more than 5 axes have been entered in the program, all (from the 6th defined axis) must be traversed in REPOS mode. The TO, the current D No. or the ZO may not be modified prior to NC START.

Bit 1: **Bit 1 = 1** The FAST BLOCK CHANGE BY FIFO function is activated for this channel (see Section 11.13 for a more detailed description).

Bit 7: Setting this bit reduces the internal running time and thus the compensating chuck withdrawal with thread tapping. The infeed axis reacts more quickly to the spindle actual-value change at the root of the thread.

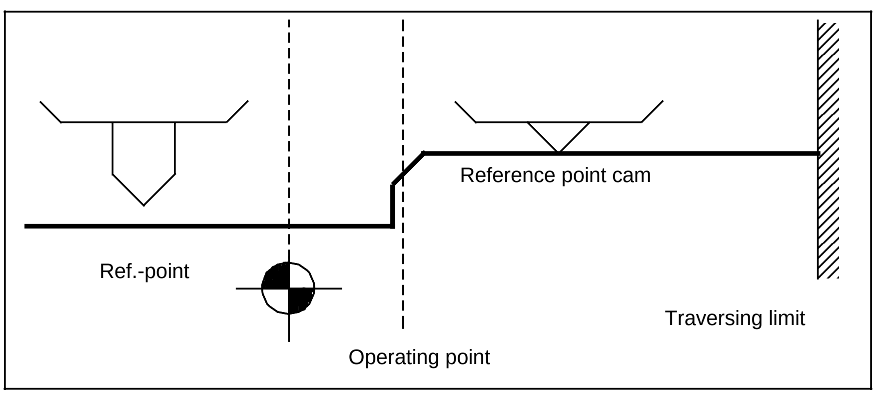
560*			NC MD Bit No.				560*
7	6	5	4	3	2	1	0
Display act. value modulo 360°	Automatic ref. point approach	Software limit switch active	No start inhibit for ref. point	Rounding for rotary axes	Rounding to whole/half degrees		No measuring circuit monitoring

Active: **When all channels of the mode group are in STOP state (except bit 5, see next page)**

Bit 7: For rotary axes only (NC MD 564* bit 5=1)!
The actual-value display jumps from 359.999 to 0 degrees after one revolution of the rotary axis.
For rotary axes also see Section 11.5.

Bit 6: **Bit 6 =0** If the axis is between the reference point cam and hardware limit switch after the controller has been switched on, the axis moves to the hardware limit switch during approach to the reference point since the controller cannot detect from the "*Deceleration" interface signal whether the axis is ahead of or behind the reference point cam (also see Section 11).

Bit 6 =1 The controller 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 (also see Section 11).



Bit 5: **Bit 5 = 0** The software limit switches in MD 224* to 236* are overrun without any response.

Active: After reference point approach

Bit 4: The program can be started without approach to the reference point for **this** axis with NC START.

With NC MD 5004 bit 3 it is possible to define for all axes together whether a reference point approach is necessary before program start.

Bit 3, 2: Active for rotary axes only! In jog mode (JOG, INC), rounding to whole or half degrees is performed (positioning) as a function of bit 2.
Bit 2 = 1 means rounding to whole degrees.

Alarm 2064 is displayed in the AUTOMATIC/MDA modes for programmed positions with no movement to half or whole degrees (rounding on rotary axis incorrectly programmed). With all axis-/spindle-specific and channel-specific alarms that effect interruption of position control (cancellation of mode group ready) and with emergency off, the control is no longer able to position the rotary axis at positions described by half or whole degrees. In such cases, the rotary axis must not be lowered into the Hirth tooth system.

Bit 0: Alarm 132* is disabled. The cables to the encoder are no longer monitored for breaks. Failure of the encoder or a wire-break is then not signalled immediately but after a delay as alarm 104*, 112* or 116* (if it is possible at all).

564*	NC MD						564*
	Bit No.						
7	6	5	4	3	2	1	0
Axis exists	Fictitious axis	Position control for rotary axis	Indexing axis	Actual values division related	Actual value sign change	Setpoint sign change	Ref. point in negative direction

Active: After POWER ON

Bit 7: The set bit causes the axis to appear on the screen and the position controller and measuring-circuit monitor to be activated. For this the corresponding PLC MD 6016 to 6018, PLC MD 6116 to 6118, PLC MD 6216 to 6218, PLC MD 6316 to 6318 must be set.

Caution!

The bit only makes the axis active after the "Power On" routine, although the axis address is displayed immediately on the screen.

Note:

A maximum of 24 axes can be activated with SINUMERIK 880 GA2.

Bit 6: Fictitious axes are required for the coordinate transformation (see NC MD 5060 to 5139).

Fictitious axes have no position control (NC MD 200* irrelevant) and need no measuring circuit. For the NC MDs which still have to be set for fictitious axes see NC MD 5060 to 5139.

Fictitious axes can be programmed on active coordinate transformation and traversed in jog modes (JOG, INC ... 10000).

Note:

Fictitious axes must be linear axes!

Bit 5: The axis in question is declared a rotary axis. Position control is recorded in degrees and G70/71 for rotary axis positions is disabled. For rotary axes see also Section 11.5.

Bit 4: With bit 4 "indexing axis" a linear or rotary axis is declared to be an indexing axis and at the MDs 1104*, 1108*, 1112* and 564* bit 3 is activated.
(For the division increment from PLC also see Section 11).

Bit 3: The actual value (the actual position display) is converted into indexing positions. An indexing position of less than 1 is not possible (applies to rotary and linear axes). The actual value display in the service data is not converted into division positions.

If bit 3 is not set for indexing axis the assignment of positions to indexing positions still occurs (via command channel) but the actual value display is in mm or inches.

Bit 2: The signs of the measuring-system pulses may be reversed by resetting the bit (necessary when the axis traverses unchecked on account of an incorrect position feedback polarity).

Bit 1: Resetting of the bit produces a change in the polarity of the speed controller set voltage (necessary when the axis moves in the mechanically incorrect direction

Bit 0: **Bit 0 = 0** Start of approach to reference point with ”**+**” direction key.

Bit 0 = 1 Start of approach to reference point with ”**-**” direction key.

568*		NC MD Bit no.				568*	
7	6	5	4	3	2	1	0
	Axis name						

Active: Immediately

The axis name is to be specified with reference to the table.

7	6	5	4	3	2	1	0	Bit No.

Extended address (address No.)	Adress 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
8 = 1000	W = 1000
9 = 1001	Q = 1001
10 = 1010	E = 1010
11 = 1011	
12 = 1100	
13 = 1101	
14 = 1111	
15 = 1111	

Example: 0000 0010=Z
0001 1001=Q1

The names in MD 5000, MD 5001 and MD 568* must not coincide. The same address name with a differently extended address is not considered to be coinciding.

Permissible names for axes, angles, chamfers and radius

A	*Unassigned address*	N	Subblock
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	Feed	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*

572*	NC MD						572*
	Bit No.						
7	6	5	4	3	2	1	0
		No distance to go calculation with JOG	Trav. the rot. axis mod. 360°	Mirroring of TO with facing axis	Rotary axis prog. modulo 360°	Plane axis	Auxiliary axis

Active: Immediately

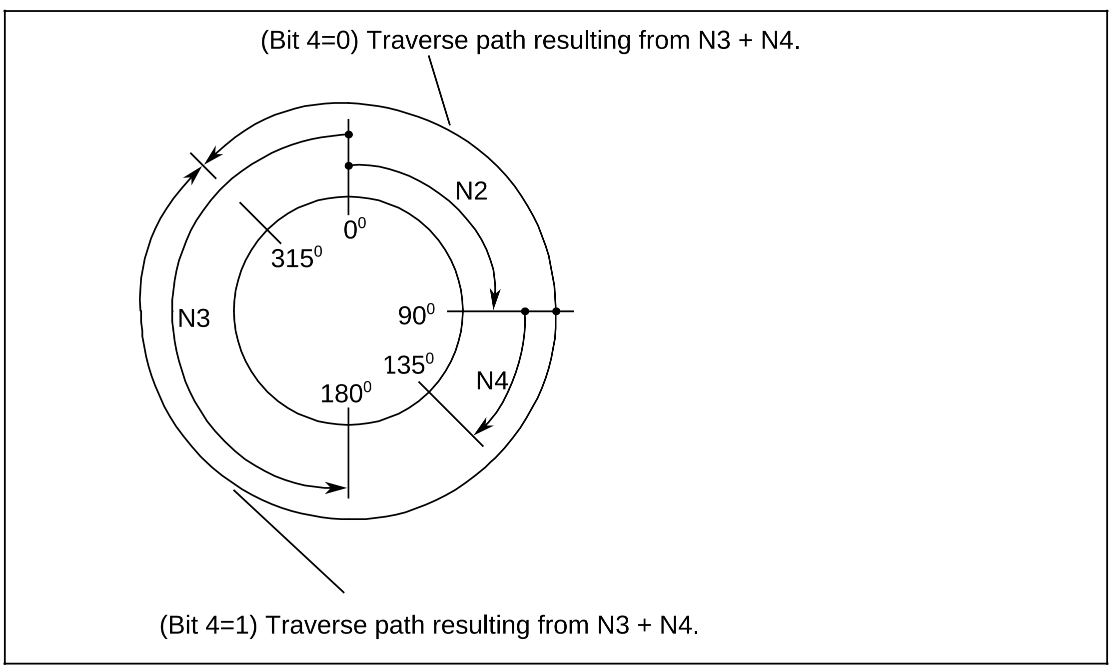
Bit 5: **Bit 5 = 0** In JOG the distance to go is calculated so that the deceleration ramp lies in front of the software limit switch, preventing the switch from being tripped.

Bit 5 = 1 The deceleration ramp is not initiated until the software limit switch is reached, so that the axis comes to a standstill behind the switch (as a function of the feedrate).

Bit 4: **Bit 4 = 0** With rotary axes on programming G90 or G68, traversing paths of > 360° or > 180° can arise and the determined direction of travel can change if zero offsets, tool offsets etc. are included in the calculation for the first time or if zero offsets, tool offsets etc. are modified within the program.

Example:

N1 G90 C0 LF (C ... rotary axis)
N2 C90 LF
N3 G58 C-180 LF
N4 G90 C+135 LF



Bit 4 = 1: In this way on G90 programming a traverse path and a direction of travel are set which correspond to the programming. Depending on the programmed G function the control behaves as follows:

G91: No modulo 360° calculation takes place. The current ZO and TO are added to the programmmed position (> 360° also possible) to give the new position and new direction of travel.

G90: The current ZO and TO are added to the programmed position (± 359.999 degrees), a modulo 360° calculation is performed and the traverse path determined in this way is traversed with the programmed direction of travel (+ ... clockwise, - ... counterclockwise) (see above example).

G68 The current ZO and TO are added to the programmed position (0 to 359.999 degrees), a modulo 360° calculation and a display to +/- 180° performed. Only in this way is the shortest direction of travel determined by the control.

Bit 3: With this function, the tool length compensations and the direction of the tool point are mirrored with tool types 1 to 9 (turning tool) depending on an axis-specific bit.

Bit 2: **Bit 2 = 1** The rotary axis can be programmed in absolute dimensions (G90) up to ± 359.999° max. The programmed sign indicates the direction of travel. (For programming rotary axes see Programming Guide).

A rotary axis can be programmed with G68 and positioning is performed over the shortest path (max. ± 180 degrees).

If a rotary axis is programmed in the part program for the first time, the control selects G68 independently of the programmed G function (G68, G90, G91). The automatic selection of G68 can only be selected with G91 CO (C ... rotary axis). The automatic selection of G68 is performed after block search or automatic block search as well and can then only be deselected by traversing the axis in REPOS. If a "modulo" rotary axis is switched to follow-up mode, NC Start automatically activates G68 for the next traversing block of this rotary axis.

If mixed programming of G90/G91 is selected in the block (NC MD 5007 bit 5) the G90/G91 function programmed last is always active for the programmed rotary axis position.

For rotary axes also see Section 11.5.

Bit 2 = 0 Travel programming and rotary axis traversing are performed as for linear axes.

Bit 1: The bits set in NC-MD 5011 act on the corresponding axis.

Bit 0: If an axis has been defined as an auxiliary axis, no TOs are added to it.

576*		NC MD Bit No.				576*	
7	6	5	4	3	2	1	0
	Axis inhibited for channel						
8	7	6	5	4	3	2	1

Active: In next block

580*		NC MD Bit No.				580*	
7	6	5	4	3	2	1	0
	Axis inhibited for channel						
16	15	14	13	12	11	10	9

Active: In next block

Bit 7, 6, 5, 4, 3, 2, 1, 0:

The axis cannot be processed by the corresponding channel if the bit is set. In case of error, alarm 3014 "Axis blocked in the channel". This disable is only active in AUTOMATIC and MDI-AUTOMATIC modes (also via command channel with path dimension/division increment from PLC).

1100*	Prelimit switch		1100*
Standard value	Lower input limit	Upper input limit	Units
20 000	**- 99 999 999**	**99 999 999**	**units (MS)**

Active: Immediately

Defines the distance at which the braking operation is to be prematurely begun if the current speed exceeds the speed in NC MD 1, thus ensuring that the position of the software limit switch will be overrun only to a negligible degree during circular interpolation.

Overshooting of the prelimit switch triggers alarm 2034 except in the case of rapid traverse.

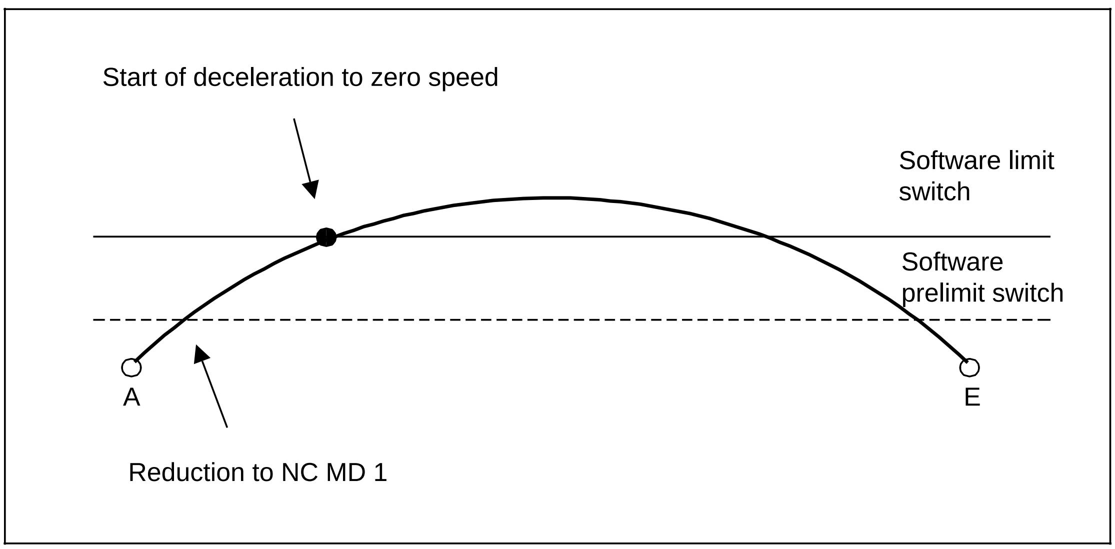
Recommendation:

The value entered should be slightly higher than the value which would correspond to the braking distance from rapid traverse to NC MD 1.

In program mode, travel movements which would result in overshooting of the software limit switch position are simply not started (alarm 2065 is triggered).

Exception:

Circular interpolation, helical interpolation



1104*	Number of divisions		1104*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**999**	**-**

Active: Immediately

Input limits: Value 0 is **not** permitted for indexing axes (NC MD 564* bit 4 = 0).

The division number defines the number of divisions of each division absolute dimension (NC MD 1108*).

(Division dimension from PLC see also Section 11).

1108*	Division reference dimension		1108*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**99 999 999**	**units (MS)**

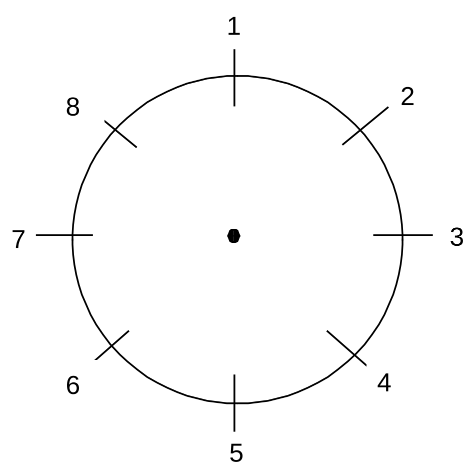
Active: Immediately

Input limits: Value 0 is **not** permitted for indexing axes (NC MD 564* bit 4 = 0).

The division reference dimension defines the reference path to which the reference number (NC MD 1104*) refers (division dimension from PLC see also Section 11).

If the indexing axis is a rotary axis, the division reference dimension is set to 360° in the control. The value entered in NC MD 1108* has no meaning.

Example:

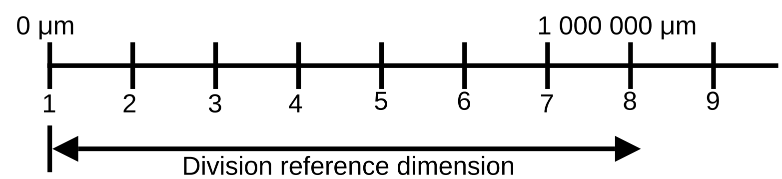


Rotary axis
Division reference dimension: 360°
Number of divisions: 8

If the indexing axis is a linear axis, the division reference dimension must be input as a reference path in units (MS) in the NC MD.

Example:

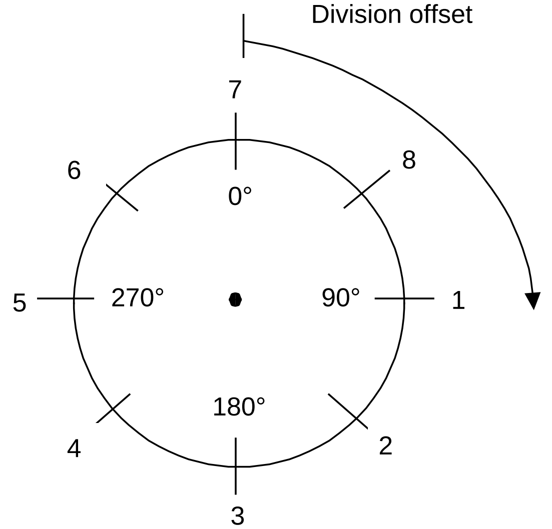
Linear axis
Division reference dimension: 1 000 000 units (MS)
Number of divisions: 8
(position control resolution): 0.5 x 10^{-3} mm



1112*	**Division offset**		**1112***
Standard value	Lower input limit	Upper input limit	Units
0	**- 99 999 999**	**99 999 999**	**units (MS)**

Active: Immediately

For the purpose of the division calculation, the indexing position 1 is made equal to 0 in the control. As these values are not the same in many cases, the reference point can be offset. This is carried out with NC MD "Division offset", which defines the distance between the actual value 0 and the division dimension 1 in units (MS).
(Division dimension from PLC see also Section 11).



Rotary axis
Division reference dimension: 360°
Number of divisions: 8
Division offset: 90000 units
(Position control resolution: 0.5 x 10^{-3} degrees)

Note:

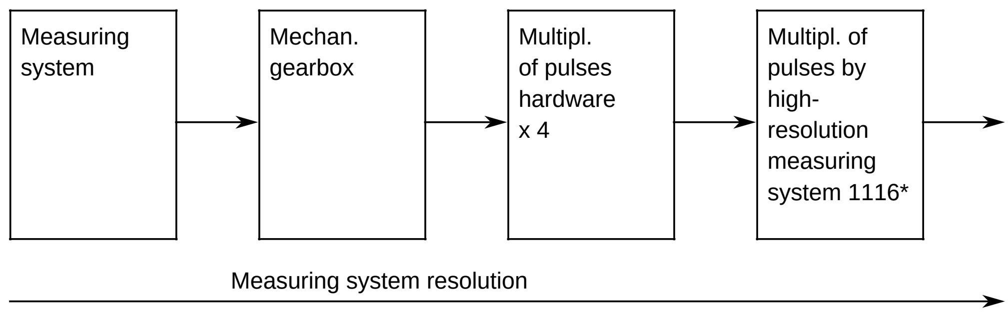
The division offset for rotary axes must be max. ± 360 degrees.

1116*	Pulse multiplication selection HMS		1116*
Standard value	Lower input limit	Upper input limit	Units
1	1	128 1)	-

Active: After POWER ON

MD 1116* is used to set the fine information for the actual position pulses when using the 6FX 1145-6B ... HMS measuring-circuit module. On Power On, the NC software checks whether an HMS measuring-circuit module has been inserted; only when this is the case does MD 1116* take effect.

Measuring system resolution : measuring system x 4 x MD 1116*



If the indexing axis is a linear axis, the division reference dimension must be input in the NC MD as a reference path in units (MS).
The set multiplication must be taken into account with variable increment weighting (NC MD 364*, 386*).

1124*	D-component feedforward control		1124*
Standard value	Lower input limit	Upper input limit	Units
0	0	1000	0.1%

Active: With activation of feedforward control

Modifies set speed proportional to the acceleration.

1) Input values: 1, 2, 4, 8, 16, 32, 64 and 128

1200*	Time delay contour monitor		1200*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**16 000**	**ms**

Active: When all channels of the mode group are in STOP state

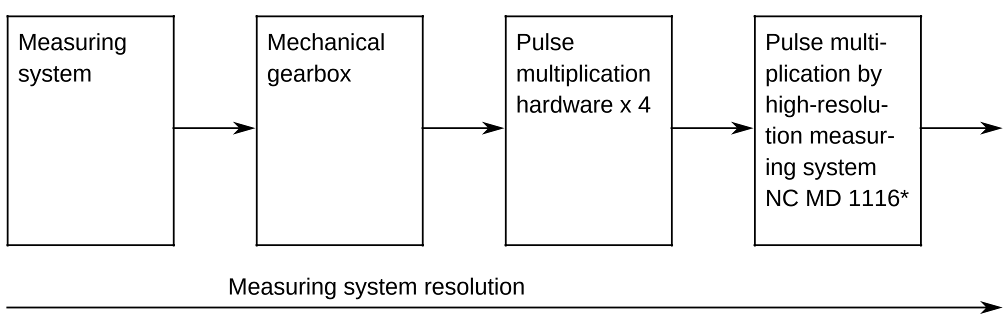
If the difference between the measured and pre-calculated following error exceeds the tolerance band MD 332* for longer than the time set in MD 1200*, the control of the axis concerned is shut down and the "contour monitor" alarm message is output.

1204*	Pulse multiplication HMS 2nd measuring system		1204*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**128**	**-**

Active: After POWER ON

With the HMS measuring circuit module, the resolution of the measuring system can easily be matched to any application.

The maximum pulse multiplication is as follows:



Multiplication factors 1, 2, 4, 8, 16, 32, 64 and 128 can be set with MD 1204*. The set multiplication must be taken into account with the variable increment weighting for the 2nd measuring system, MD 1208*, MD 1212*.

1208*	Pulses variable increment weighting 2nd measuring system		1208*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**65 000**	**-**

Active: After POWER ON

Separate increment weighting can be selected for the second position measuring system. The meaning corresponds to that of MD 364* for the first measuring system.

1212*	Traversing path variable increment weighting 2nd measuring system		1212*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**65 000**	**-**

Active: After POWER ON

Separate increment weighting can be selected for the second position measuring system. The meaning corresponds to that of MD 368* for the first measuring system.

1216*	Changeover tolerance 2nd measuring system		1216*
Standard value	Lower input limit	Upper input limit	Units
1 000	**0**	**16 000**	**units (MS)**

If a second measuring system is entered in MD 1388* and defined in MD 1824* bit 1, a switch can be made from the first to the second measuring system with the interface signal "measuring system 1/2".

A tolerance band within which the divergence between the actual values of the two measuring systems must lie can be specified with MD 1216*.

If the divergence is greater than the tolerance, no switch is made and the "measuring circuit error" alarm is issued.

1224*	Cutout delay servo enable		1224*
Standard value	Lower input limit	Upper input limit	Units
400	0	1 000	ms

Active: After POWER ON

The speed enable (servo enable) at the measuring circuit is cancelled after the set delay. The controller enable exists on the measuring circuit once per axis/spindle and is initialized specific to the mode group by the control.

The entered time delay has the following effects:

1. Once the interpolator has reached the programmed position, the clamping tolerance (MD 212*) is activated when this time delay has expired. At this instant the following error must therefore be smaller than the clamping tolerance. The time must be set long enough to allow the maximum following error (rapid traverse) to be cancelled. In the event of an error, the servo enable at the measuring circuit is cancelled and alarm 112* (zero speed monitor) is issued.

   This applies only if NC MD 372* (delay zero speed monitor) is at 0.
2. Delay before cancelling the servo enable at the measuring circuit after EMERGENCY STOP and other errors that lead to an immediate standstill of the axes (e.g. contour monitoring).
3. Delay before cancelling the servo enable at the measuring circuit if the PLC cancels the servo enable for one axis.

1228*	Backlash compensation 2nd measuring system		1228*
Standard value	Lower input limit	Upper input limit	Units
0	-16 000	16 000	units (MS)

The MD 1228* (backlash compensation 2nd measuring system) has the same meaning/unit as MD 220* (backlash compensation 1st measuring system).

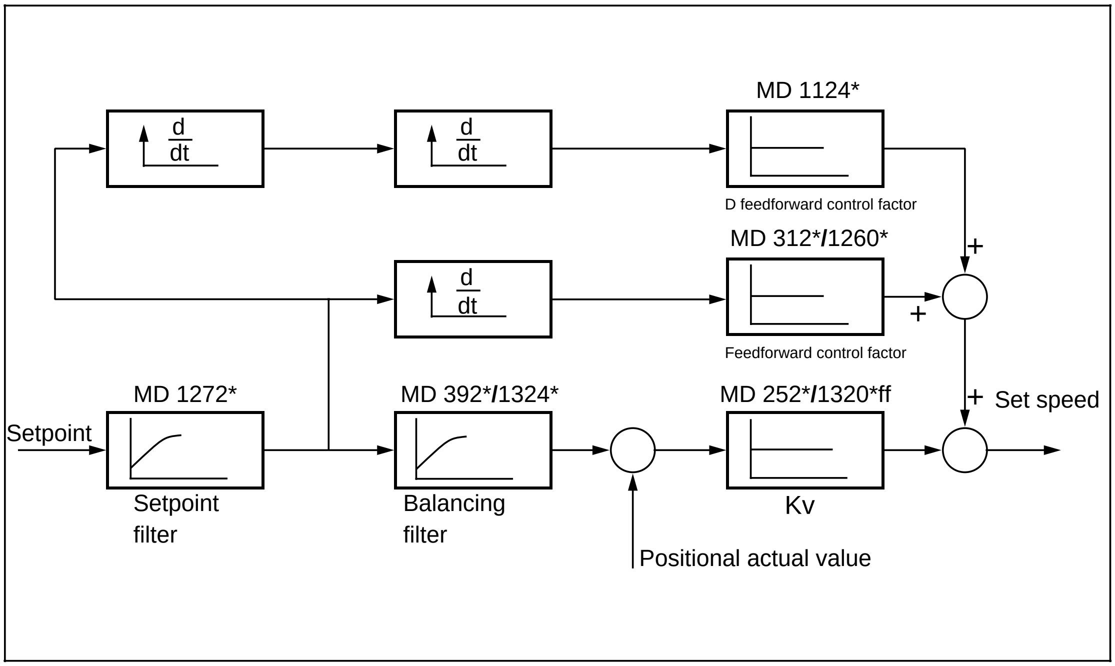
1260*	Feedforward control factor for tapping without compensating chuck		1260*
Standard value	Lower input limit	Upper input limit	Units
0	0	1 000	0.001

A P-component feedforward control factor can be entered for rotary axis operation with the function "tapping without compensating chuck". The meaning corresponds to that of NC MD 312*.

1272*	Time constant setpoint smoothing		1272*
Standard value	Lower input limit	Upper input limit	Units
0	0	1 000	0.1 ms

Active: Immediately

The setpoint filter prevents overshooting of the axis position with dynamic speed feedforward control (NC MD 312*) of 100 %.

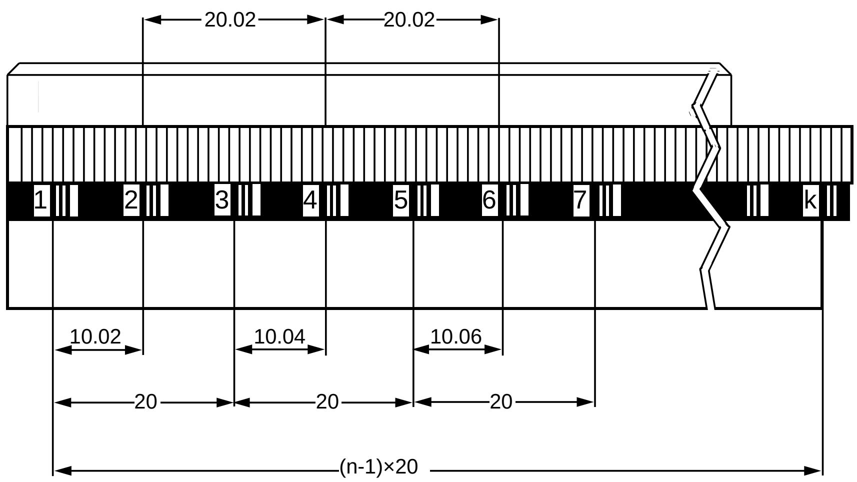


Setpoint smoothing has to be activated with NC MD 1820* bit 0.

1300*	Basic spacing distance-coded reference marks		1300*
Standard value	Lower input limit	Upper input limit	Units
1 000	**0**	**16 000**	**see below**

Active: After POWER ON

When using linear scales with distance-coded reference marks, the basic spacing of the reference marks has to be entered here.



The basic spacing used here is 20 mm.

The basic spacing is stated in multiples of the division period (grating distance, e.g. 20 µm).

 6ZB5 410-0HG02

1304*	External pulse multiplication (EXE)		1304*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**100**	**-**

External pulse multiplication with linear measuring systems. Number of signal periods (marks) generated by external interpolation and digitizing electronics from one division period of the linear scale (grating distance).

This MD is required for the "distance-coded reference marks" function.

1320*	Servo gain factor for thread tapping without compensating chuck		1320*
Standard value	Lower input limit	Upper input limit	Units
1 666	**0.166**	**10 000**	**0.01 s^{-1}**

Active: When all channels of the mode group are in STOP state

This NC MD is comparable to NC MD 252*. However, it applies only to rotary axis operation with the function "tapping without compensating chuck" (Section 11).

Since the rotary axis and the infeed axis interpolate in the case of tapping, it is essential that the effective servo gain (K_V) factors are set the same for both these axes.

This machine data modifies the servo gain (K_V) factor of the linear axis. The servo gain (K_V) factor of the rotary axis remains unchanged.

To reduce the following error, the dynamic feedforward control can also be used (NC MD 312*).

Note:

Input value "0" opens the position control circuit.

1324*	Time constant for dyn. feedforward control for tapping without compensating chuck		1324*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1000**	**0.1 ms**

Active: In next block

A time constant for dynamic feedforward control can be entered for the rotary axis with the function "tapping without compensating chuck".

This machine data has the same meaning as NC MD 392*.

1388*	Axis assignment 2nd measuring system		1388*
Standard value	Lower input limit	Upper input limit	Units
see below	**0**	**10 030 000**	**-**

Active: After POWER ON

In the case of NC axes that are equipped with a 2nd position measuring system (e.g. linear scale), an actual-value assignment corresponding to the assignment for the 1st measuring system in NC MD 200* has to be made with MD 1388*.

The measuring circuits do not have to be assigned to the relevant servo CPU as this has already been done in NC MD 200*.

:	:	:	:
Measuring circuit module No.	No. of encoder (measuring circuit)	No significan-ce	No significan-ce

1396*	Ratio of position control cycle to servo cycle		1396*
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**64**	**-**

Active: After POWER ON

In the basic setting, all the axes of a servo CPU have a set position control cycle that corresponds to the servo cycle time. The servo cycle time is calculated as follows:

Servo cycle time = System clock * MD 16* Possible inputs: 1, 2, 4, 8, 16, 32, 64

x = 0 for servo CPU 1
1 for servo CPU 2
2 for servo CPU 3
3 for servo CPU 4

The position control cycle can set axis-specifically with MD 1396* as follows:

Position control cycle = Servo cycle time * MD 1396*

Condition: $\frac{\text{Interpolation time}}{\text{Position control cycle}} = \text{Integer}$

1800*	NC MD Bit No.						1800*
7	6	5	4	3	2	1	0
Display resolution				Position control resolution			

Active: After POWER ON

For an exact description and conditions (axis-specific revolutions) see Section 11.7. Coding of the axis-specific resolutions (input resolution, display resolution, position control resolution)

Table of resolution codes

Bit 7	Bit 6	Bit 5	Bit 4	Input resolution			NC MD 5002
Bit 7	Bit 6	Bit 5	Bit 4		Display resolution		NC MD 1800*
Bit 3	Bit 2	Bit 1	Bit 0			Pos. contr. resol.	NC MD 1800*
0	0	0	0	———	10^{-1} [mm] [degr.]	0.5 x10^{-1} [degr.]	
1	0	0	0	10^{-2} [mm] [degr.]	10^{-2} [mm] [degr.]	0.5 x 10^{-2} [degr.]	
0	1	0	0	10^{-3} [mm] [degr.]	10^{-3} [mm] [degr.]	0.5 x 10^{-3} [degr.]	
1	1	0	0	———	———	2 x 10^{-4} [degr.]	Metric (degrees)
0	0	1	0	10^{-4} [mm] [degr.]	10^{-4} [mm] [degr.]	0.5 x 10^{-4} [degr.]	
1	0	1	0	10^{-5} [mm] [degr.]	10^{-5} [mm] [degr.]	0.5 x 10^{-5} [degr.]	
0	1	1	0	———	———	———	
1	1	1	0	———	———	———	
0	0	0	1	———	10^{-1} [degr.]	0.5 x 10^{-1} [degr.]	
1	0	0	1	———	10^{-2} [degr.]	0.5 x 10^{-2} [degr.]	
0	1	0	1	10^{-3} [inch] [degr.]	10^{-3} [inch] [degr.]	0.5 x 10^{-3} [inch] [degr.]	
1	1	0	1	10^{-4} [inch] [degr.]	10^{-4} [inch] [degr.]	0.5 x 10^{-4} [inch] [degr.]	Inch (degrees)
0	0	1	1	———	———	2 x 10^{-5} [inch]	
1	0	1	1	10^{-5} [inch] [degr.]	10^{-5} [inch] [degr.]	0.5 x 10^{-5} [inch] [degr.]	
0	1	1	1	10^{-6} [inch] [degr.]	———	———	
1	1	1	1	———	———	———	

0 1 0 0 = standard machine data

1804*		NC MD Bit No.				1804*	
7	6	5	4	3	2	1	0
						Autom. tacho com-pensation ON	

Active: Immediately

Bit 1: "Autom. tacho compensation ON"

0 signal: Tacho compensation is not active for the axis concerned.

1 signal: Tacho compensation is active for the axis concerned.

This function calculates a direction-dependent compensation value for axes traversing at constant velocity with reference to the following error model of the contour monitor (cf. NC MD 332*). This value can compensate for deviations of up to 12 % of the multgain (multgain fine adjustment) and is applied parallel to the P-component feedforward control. The tacho compensation thus automatically activates the "feedforward control" internally (note computing time requirement). If a new compensation value cannot be calculated (e.g. no constant velocity traversing, 12 % limitation, velocity less than $1/8 \cdot N_{max}$), the compensation value last identified as useful remains in use (self-learning compensation). A separate compensation value is calculated for each direction of travel in order to equalize asymmetries. The compensation value is deleted when compensation is deselected. If the tacho compensation is to be used for spindles as well (e.g. pair of synchronous spindles), a C axis must be assigned to each spindle first in order to allow the axis-specific MD bit to be set.

1808*		NC MD Bit No.				1808*	
7	6	5	4	3	2	1	0
		Measuring system 2 distance-coded	Measuring system 1 distance-coded	Absolute offset valid	Abs. enc. feedback polarity opposite to machine system		SIPOS absolute encoder available

Bit 0=1 The NC axis is equipped with a SIPOS absolute encoder (see Section 11.18 for explantion).

Bit 2=1 For calculating the relevant machine absolute value after NC Reset it must be known whether the value of the absolute encoder increases or decreases with the machine absolute value. Bit 2=1 means that the value of the absolute encoder increases with decreasing machine absolute value or decreases with greater machine absolute value.

Bit 2=0 Machine system and SIPOS absolute system or distance-coded reference system have the same feedback polarity.

Bit 3=1 The value for absolute value offset contained in NC MD 396* is valid.

Meaning with SIPOS: After POWER ON, "reference point reached" is set for this axis.

Meaning with distance coding: The referencing procedure is executed with this value and acknowledged with "reference point reached".

Bit 4=1 Measuring system 1 is equipped with a linear scale with distance-coded reference marks. **Active: After POWER ON.**

Bit 5=1 Measuring system 2 is equipped with a linear scale with distance-coded reference marks. **Active: After POWER ON.**

The following NC machine data must be considered if the SIPOS absolute encoder or distance-coded reference marks are used:

Axial machine data:

MD 240* reference point ordinates
MD 396* absolute offset

MD 1300* Basic spring of reference marks
MD 1304* External pulse multiplication (not with SIPOS) absolute encoders)

Note:

Bit 2 and 3 become active immediately after a change, that is to say with the next reference point approach.

Procedure for reference point approach

When MD 1808* bit 0 "Axis with SIPOS absolute encoder" is set, a distinction is made between two different conditions.

Condition 1: MD 1808* bit 3=0

When the bit "absolute offset valid" is not set, reference point approach is executed as for an axis without an absolute encoder. On "reference point reached" the calculated absolute offset is transferred to the axial MD 396* and MD 1808* bit 3, "absolute offset valid" is set for the relevant axis.

The absolute offset is calculated according to the equation:

machine system = SIPOS system + absolute offset

or

absolute offset = machine system - SIPOS system

where the following applies:

machine system = desired absolute position = reference point ordinate

and

SIPOS system = displayed absolute position (actual position)

Note:

If a value other than 0 has been entered in MD 396*, this value must be considered because it is contained in the SIPOS system (= displayed actual value).

Condition 2: MD 1808* bit 3=1

If the "absolute offset valid" bit is set, reference point approach is suppressed. This is valid until the user resets the bit.

Note:

If function G74, "Reference point approach from part program" is programmed, referencing is not executed for the second condition and the program continues from the subsequent block.

Behaviour after warm restart (POWER ON)

If MD 1808* bit 0, "Axis with absolute encoder", is set, MD 1808* bit 3, "Absolute offset valid" is checked for the axis in question. If this bit has also been set, the axial interface signal "Reference point reached" is set on warm restart.

Special case - "Parking axis"

The axis interface signal, "Parking axis" also deletes the interface signal "Reference point reached" for an axis with SIPOS absolute encoder. If MD 1808* bit 3 is set, a renewed reference point approach is suppressed. The absolute value is not adopted until POWER ON.

Note:

Reference point approach can be executed again when MD 1808* bit 3 has been deleted.

1820*		NC MD Bit No.				1820*	
7	6	5	4	3	2	1	0
Contour monitoring OFF	Pulse coder monitoring ON					Zero monitoring ON	Setpoint smoothing ON

Bit 0: Setpoint smoothing ON

Bit 0=1 The smoothing filter in the setpoint path is active. It is needed to equalize dynamic differences between interpolating or ELG-linked axes. The time constants of the setpoint filters can be entered in NC MD 1272* and 486*.

Bit 0=0 Setpoint smoothing deselected

Bit 1: Zero smoothing ON

Bit 1=1 Monitoring determines whether pulses are lost between two zero mark passes.
Where appropriate, alarm 144* is triggered (also see the description of alarm 144*).

Bit 1=0 Monitoring deselected

Bit 6: Pulse coder monitoring ON

Bit 6=1 Monitoring determines how many direction reversals took place between two measuring operations. If too many direction reversals are ascertained, alarm 140* is triggered (also see the description of alarm 140*).

Bit 6=0 Monitoring deselected

Bit 7: Dynamic contour monitoring

Bit 7=1 Dynamic monitoring deselected.

Bit 7=0 Dynamic monitoring selected.

1824*		NC MD Bit No.				1824*	
7	6	5	4	3	2	1	0
Assignment leadscrew error compen-sation			Partial actual value inversion 2nd measuring system			2nd measuring system available	

Bit 7: With the aid of this bit the leadscrew error compensation can be assigned to the 1st or 2nd measuring system.

Bit 7=1 Leadscrew error compensation assigned to 2nd measuring system.

Bit 7=0 Leadscrew error compensation assigned to 1st measuring system.

Bit 4: The sign of the measuring system pulses can be inverted by setting/resetting this bit.

Bit 1: A 2nd measuring system for an axis can be activated by setting this bit.

Bit 1=1 Actual values of the 1st or 2nd measuring system are evaluated (depending on DB 32 DW (C + 2) bit 12).

Bit 1=0 Actual values of the 1st measuring system are evaluated.

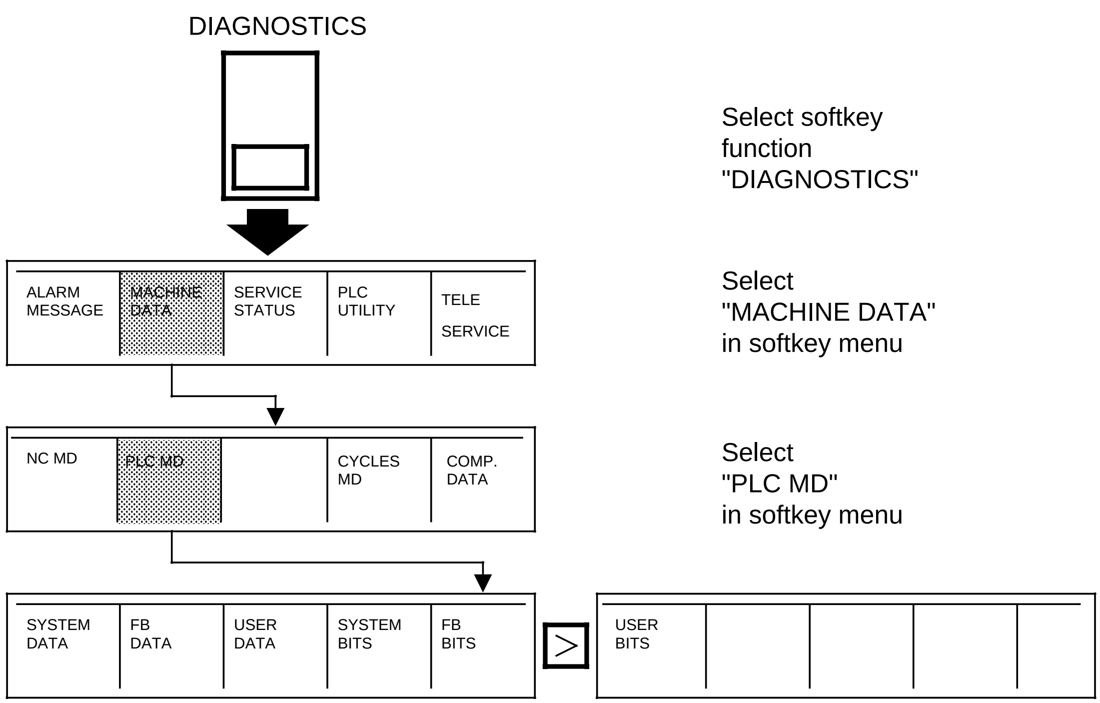
Active: After POWER ON

9.4 PLC machine data

9.4.1 General

The PLC machine data are only active after "POWER ON Reset". Inputting of the PLC machine data via the serial interface is possible in the "OVERALL RESET" installation mode. Manual input of PLC-MD or editing of PLC-MD is only possible in "NORMAL" installation mode.

PLC MD for PLC1	PLC MD for PLC2	
0 to 199	200 to 399	PLC MD for operating system
2000 to 2149	2250 to 2399	PLC MD for function blocks (Siemens)
4000 to 4049	4100 to 4149	PLC MD for user
6000 to 6099	6100 to 6199	PLC MD bits for operating system
6400 to 6699	6400 to 6699	General bits for both PLCs
7000 to 7249	7250 to 7499	PLC MD bits for function blocks (Siemens)
8000 to 8049	8050 to 8099	PLC MD bits for user



Select softkey function "DIAGNOSTICS"

Select "MACHINE DATA" in softkey menu

Select "PLC MD" in softkey menu

9.4.2 PLC MD description

9.4.2.1 PLC machine data for the operating system

2	Time call reference of OB 5		2
Standard value	Lower input limit	Upper input limit	Units
1	1	3	2.5 ms

Here the user defines the time call reference n*2.5 ms by inputting the factor n=1, 2, 3.

3	Time call reference of OB 6		3
Standard value	Lower input limit	Upper input limit	Units
1	1	9	10 ms

The user defines the time call reference m*10 ms by inputting the factor m = 1, ... 9.

4	Time call reference of OB 7		4
Standard value	Lower input limit	Upper input limit	Units
1	1	255	100 ms

The user defines the time call reference m*1000 ms by entering the factor p = 1, ... 255.

5	Last STEP 5 timer		5
Standard value	Lower input limit	Upper input limit	Units
64	**-1**	**+255**	**-**

The processing of individual timers places a certain load on the PLC operating system. If in the user program, for example, only the timers 1 to 20 are required, input the number 20 and the system program will only process these timers. This saves processing time and a better PLC cycle time can be achieved.

The input of "–1" means that **all** timers are disabled.
Times T128 to T255 are not available until Option N05 is activated.

6	1st input byte for operator panel inputs		6
Standard value	Lower input limit	Upper input limit	Units
64	**0**	**96**	**-**

The user defines from which input byte the operator panel inputs are located (minimum value = IB 64, maximum value = 96).

7	1st output byte for operator panel outputs		7
Standard value	Lower input limit	Upper input limit	Units
64	**0**	**112**	**-**

The user defines from which output byte the operator panel outputs are located (minimum value = QB 64, maximum value = 112).

8	Last active channel		8
Standard value	Lower input limit	Upper input limit	Units
16	**1**	**16**	**-**

Here the last activated channel (channel number) is communicated to the PLC system program.

9	Last active spindle		9
Standard value	Lower input limit	Upper input limit	Units
6	**1**	**6**	**-**

Here the last spindle to be activated (spindle number) is communicated to the PLC system program.

10	Last active axis		10
Standard value	Lower input limit	Upper input limit	Units
24	**1**	**24**	**-**

Here the last axis to be activated (axis number) is communicated to the PLC system program.

11	Erase limit input image		11
Standard value	Lower input limit	Upper input limit	Units
127	**63**	**127**	**-**

The user can define up to which input byte the input image will be erased on warm restart. This means that the input bytes which are not reserved by process I/Os or I/O modules can be used as additional flag bytes.

Example: Erase limit PLC MD 11 = 72

I byte 0
:
:
:
I byte 63 — Signals from the machine

I byte 64
:
:
:
I byte 72 — Reserved by 2 I/O modules

I byte 73
:
:
I byte 127 — Area which can be used as a free flag area

12	Erase limit output image		12
Standard value	Lower input limit	Upper input limit	Units
127	**63**	**127**	**-**

The user can define up to which output byte the output image will be erased on warm restart. This means that the output bytes which are not reserved by process I/Os or I/O modules can be used as additional flag bytes.

17	Number of waiting cycles for an assigned user interface		17
Standard value	Lower input limit	Upper input limit	Units
1	**1**	**10**	**-**

The parameterized delay is specified as the number of PLC cycles to wait for the user interface to be enabled and is valid for all user interfaces of a PLC. After the delay has elapsed a message frame for the user interface is acknowledged negatively.

Standard value : 1 i.e. after one PLC cycle the user interface must be free again.
max. input value : 10 i.e. after 10 PLC cycles the user interface must be free again.

18	Number of the user interface which is processed on synchronization		18
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**31**	**-**

Message output to the whole computer during synchronization is only possible for the user interface specified here.

Standard value : 0, i.e. processsing of first user interface possible during synchronisation as well
max. input value : 31

19	Number of the function Nos.for kernel sequence initiation from UI		19
Standard value	Lower input limit	Upper input limit	Units
3	**0**	**10**	**-**

Number of function Nos. for a kernel sequence initiation from user interface.

Input values: 0 = Kernel sequence initiation from user interface not permitted in this PLC
1...10 (max.) = Kernel sequence initiation from user interface permissible. The number of function Nos. starting with the machine data DB 60, DW 20 is specified for which a kernel sequence initiation from user interface occurs.
3 = Standard value

20-29	**Function No. for kernel sequence initiation from user interface**		**20-29**
Standard value	Lower input limit	Upper input limit	Units
see below	**0**	**255**	**-**

Function Nos. which initiate a kernel sequence via user interface.
The area must be assigned continuously from DB 61, DW 20.

Standard value	MD 20	:	Fct. No.	25	(Message R_TS__)
Standard value	MD 21	:	Fct. No.	26	(Message R_WB__)
Standard value	MD 22	:	Fct. No.	30	(Message R_WE__)
Standard value	MD 23	:	Fct. No.	0	
:	:				
:	:				
:	:				
Standard value	MD 29	:	Fct. No.	0	
Max. input value	. . .	:	Fct. No.	255	

33	**Number of user interfaces for command channel**		**33**
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**8**	**-**

Here the number of user interfaces is communicated to the PLC system program in DB 41.

34	Initial address 1st DMP submodule 1st DMP interface 1st line		34
Standard value	Lower input limit	Upper input limit	Units
-1	**0**	**254**	**-**

⋮

48	Initial address 15th DMP submodule 1st DMP interface 1st line		48
Standard value	Lower input limit	Upper input limit	Units
-1	**0**	**254**	**-**

- These MDs in the area of the specific PLC operating system MD values are used to define the initial address (byte number) for each connected DMP station.
- This initial address is valid with connected DMP stations for both inputs and outputs.
- These machine data also declare the DMP stations that are available for operation. if the relevant MD is set to -1, the associated station is declared not available. This is the default setting of the machine data. Values greater than -1 up to the maximum value of 254 define the byte numbers stated above for the stations.
- The machine data are checked for permissibility (e.g. coincidence) by the collaboration of the PLC and DMP interface. The PLC detects errors and branches to the STOP state.
- Only the declared stations are operated and monitored by the PLC and Interface DMP.
- These MDs are ineffective with "free programming"; also see MD 136.
- See Section 7.6 for a detailed description.

PLC MD, DB60 DW	DMP IM	MPC	DMP station	PLC MD default value	DMP rotary switch position
34	1	1	1	- 1	E
35	1	1	2	- 1	D
36	1	1	3	- 1	C
37	1	1	4	- 1	B
38	1	1	5	- 1	A
39	1	1	6	- 1	9
40	1	1	7	- 1	8
41	1	1	8	- 1	7
42	1	1	9	- 1	6
43	1	1	10	- 1	5
44	1	1	11	- 1	4
45	1	1	12	- 1	3
46	1	1	13	- 1	2
47	1	1	14	- 1	1
48	1	1	15	- 1	0
49	1	2	1	- 1	E
50	1	2	2	- 1	D
51	1	2	3	- 1	C
52	1	2	4	- 1	B
53	1	2	5	- 1	A
54	1	2	6	- 1	9
55	1	2	7	- 1	8
56	1	2	8	- 1	7
57	1	2	9	- 1	6
58	1	2	10	- 1	5
59	1	2	11	- 1	4
60	1	2	12	- 1	3
61	1	2	13	- 1	2
62	1	2	14	- 1	1
63	1	2	15	- 1	0
64	2	1	1	- 1	E
65	2	1	2	- 1	D
66	2	1	3	- 1	C
67	2	1	4	- 1	B
68	2	1	5	- 1	A
69	2	1	6	- 1	9
70	2	1	7	- 1	8
71	2	1	8	- 1	7
72	2	1	9	- 1	6
73	2	1	10	- 1	5
74	2	1	11	- 1	4
75	2	1	12	- 1	3
76	2	1	13	- 1	2
77	2	1	14	- 1	1
78	2	1	15	- 1	0

PLC MD, DB60 DW	DMP IM	MPC	DMP station	PLC MD default value	DMP rotary switch position
79	2	2	1	- 1	E
80	2	2	2	- 1	D
81	2	2	3	- 1	C
82	2	2	4	- 1	B
83	2	2	5	- 1	A
84	2	2	6	- 1	9
85	2	2	7	- 1	8
86	2	2	8	- 1	7
87	2	2	9	- 1	6
88	2	2	10	- 1	5
89	2	2	11	- 1	4
90	2	2	12	- 1	3
91	2	2	13	- 1	2
92	2	2	14	- 1	1
93	2	2	15	- 1	0
94	3	1	1	- 1	E
95	3	1	2	- 1	D
96	3	1	3	- 1	C
97	3	1	4	- 1	B
98	3	1	5	- 1	A
99	3	1	6	- 1	9
100	3	1	7	- 1	8
101	3	1	8	- 1	7
102	3	1	9	- 1	6
103	3	1	10	- 1	5
104	3	1	11	- 1	4
105	3	1	12	- 1	3
106	3	1	13	- 1	2
107	3	1	14	- 1	1
108	3	1	15	- 1	0
109	3	2	1	- 1	E
110	3	2	2	- 1	D
111	3	2	3	- 1	C
112	3	2	4	- 1	B
113	3	2	5	- 1	A
114	3	2	6	- 1	9
115	3	2	7	- 1	8
116	3	2	8	- 1	7
117	3	2	9	- 1	6
118	3	2	10	- 1	5
119	3	2	11	- 1	4
120	3	2	12	- 1	3
121	3	2	13	- 1	2
122	3	2	14	- 1	1
123	3	2	15	- 1	0

124	Byte number of 1st alarm byte		124
Standard value	Lower input limit	Upper input limit	Units
-1	**0**	**127**	**-**

127	Byte number of 4th alarm byte		127
Standard value	Lower input limit	Upper input limit	Units
-1	**0**	**127**	**-**

Up to 4 alarm input bytes (for the OB2 call) can be declared. The alarm bytes can also be located on the DMP modules and expansion units.

130	No. of interrupt byte of 1st DMP interface, 1st line		130
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

131	No. of interrupt byte of 1st DMP interface, 2nd line		131
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

132	No. of interrupt byte of 2nd DMP interface, 1st line		132
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

133	No. of interrupt byte of 2nd DMP interface, 2nd line		133
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

134	No. of interrupt byte of 3rd DMP interface, 1st line		134
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

135	No. of interrupt byte of 3rd DMP interface, 2nd line		135
Standard value	Lower input limit	Upper input limit	Units
-1	**-1**	**254**	**-**

A maximum of 3 interrupt bytes can be defined with PLC MDs 130 to 135. A line cannot contain more than one interrupt byte. The interrupt inputs then have to be enabled and a signal change defined with PLC MDs 6068 to 6079.

Note:

These MDs are ineffective with "free programming" (see PLC MD 136).

136	No. of configuration data block for free programming		136
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**255 or 1255**	**-**

The person carrying out the installation can choose between two methods of I/O device assignment:

1. Fixed configuration:

The input/output bytes must be assigned to the PLC I/O devices by means of PLC MD (34 to 135 and 6068 to 6079). In this case PLC MD 136 must be 0.

2. Free configuration:

The PLC I/O devices are assigned their input/output bytes by means of an independent configuration data block. This process is also referred to as "free configuration" because different initial numbers can be used for the input and output bytes. The number of the DB or DX used must be entered in PLC MD 136.

Value range:	0	to	255	for DB0 to DB255
	1000	to	1255	for DX0 to DX255

The specified DB or DX must lie in the range to which the user has access.

Entering 0 in PLC MD 136 selects "fixed configuration".

The default setting of PLC MD 136 is "0". See Section 7.7.2 for further information.

9.4.2.2 PLC MD for function blocks
PLC - MD 2000 to 2149 for PLC 1
PLC - MD 2250 to 2399 for PLC 2

2000-2149	PLC 1		2000-2149
2250-2399	PLC 2		2250-2399
Standard value	Lower input limit	Upper input limit	Units

Here a range of 249 PLC MD words has been reserved for the adaptation of future PLC packages and function blocks. You will find the description with the PLC packages and function blocks in question.

9.4.2.3 PLC MD for user
PLC - MD 4000 to 4049 for PLC 1
PLC - MD 4100 to 4149 for PLC 2

4000-4049	PLC 1		4000-4049
4100-4149	PLC 2		4100-4149
Standard value	Lower input limit	Upper input limit	Units

Here a range of 50 PLC MD words is available for the user's use. These can be used to adapt the user machine program to the corresponding machine configuration (see also PLC MD 8000 to 8049).

9.4.3 PLC MD bits

PLC machine data for the basic program are required for two functions:

a) Adaptation of the basic program to the actual NC structure

Example of application:

The following functions must be activated:

NC functions	PLC MD
NC channel 1	6000.0
NC channel 2	6000.1
Spindle 1	6012.0
Axis 1	6016.0
Axis 2	6016.1
Axis 3	6016.2
Axis 4	6016.3
RS 232 input/output by the PLC	6026.7
Input submodule 1	6027.0
Output submodule 1	6027.4

b) Definition of the bytes for PLC error and operational messages

DB 10	Message No.			PLC MD (PLC 1)
DL 6	6000 : 6007	Error messages	Feed disable overall	6032.0
DR 6	6008 : 6015			
DL 7	6016 : 6023	Operational messages	Feed disable overall **and** read in disable	6040.2
DR 7	6024 : 6031			
DL 8	6032 : 6039			6032.4
DR 8	6056 : 6063	Error messages		
DL 9	6080 : 6087		Read in disable	
DR 9	6080 : 6087	Operational messages		6040.7
DL 10	6080 : 6087			
DR 10	6080 : 6087	Error messages		6033.1
DL 11	6080 : 6087	Error messages	Disable NC start	6033.2
DR 11	6088 : 6095			

Table of error numbers

Channel 1	6000 - 6095	DB 10, DW 6 - DW 11
Channel 2	6100 - 6195	DB 11, DW 6 - DW 11
Channel 3	6200 - 6295	DB 12, DW 6 - DW 11
Channel 4	6300 - 6395	DB 13, DW 6 - DW 11
Channel 5	6400 - 6495	DB 14, DW 6 - DW 11
Channel 6	6500 - 6595	DB 15, DW 6 - DW 11
Channel 7	6600 - 6695	DB 16, DW 6 - DW 11
Channel 8	6700 - 6795	DB 17, DW 6 - DW 11
Channel 9	6800 - 6895	DB 18, DW 6 - DW 11
.	.	.
.	.	.
.	.	.
Channel 16	7500 - 7595	DB 25, DW 6 - DW 11
Spindle 1	8000 - 8015	DB 31, DW 3
Spindle 2	8020 - 8035	DB 31, DW 7
Spindle 3	8040 - 8055	DB 31, DW 11
Spindle 4	8060 - 8075	DB 31, DW 15
Spindle 5	8080 - 8095	
Spindle 6	8100 - 8115	
Axis 1	8200 - 8215	DB 32, DW 3
Axis 2	8220 - 8235	DB 32, DW 7
Axis 3	8240 - 8255	DB 32, DW 11
Axis 4	8260 - 8275	DB 32, DW 15
Axis 5	8280 - 8295	DB 32, DW 19
Axis 6	8300 - 8315	DB 32, DW 23
Axis 7	8320 - 8335	DB 32, DW 27
Axis 8	8340 - 8355	DB 32, DW 31
Axis 9	8360 - 8375	DB 32, DW 35
Axis 10	8380 - 8395	DB 32, DW 39
Axis 11	8400 - 8415	DB 32, DW 43
Axis 12	8420 - 8435	DB 32, DW 47
Axis 13	8440 - 8455	DB 32, DW 51
Axis 14	8460 - 8475	DB 32, DW 55
Axis 15	8480 - 8495	DB 32, DW 59
Axis 16	8500 - 8515	DB 32
.	.	.
.	.	.
.	.	.
Axis 24	8660 - 8675	DB 32,
PLC 1	9000 - 9247	DB 58, DW 1 - DL 16
PLC 2	9250 - 9497	DB 58, DW 1 - DL 16

9.4.3.1 PLC MD bits for operating system (DB 63) NC MD 6000 to 6099 for PLC 1 NC MD 6100 to 6199 for PLC 2

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6000	Signals from/to NC channel							
DL 0	8	7	6	5	4	3	2	1
6001	Signals from/to NC channel							
DR 0	16	15	14	13	12	11	10	9

Bit = 0 The signals from and to the NC channel are not transferred to the channel.
Bit = 1 The signals from and to the NC channel are transferred to the channel (including M decoding M00 - M99 and output of the auxiliary functions and block information).

Note:

- The bit may only be set for one PLC as the signals to the NC channel can only be output by one PLC.
- If the bits for PLC 1 and PLC 2 are set the system branches into the stop loop (for error list see Interface Description Part 1).
- With the SINUMERIK 880 GA2 a maximum of 8 (out of 16) channels can be activated.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6003	Signals from the NC channel							
DR 1	8	7	6	5	4	3	2	1
6004	Signals from the NC channel							
DL 2	16	15	14	13	12	11	10	9

Bit = 0 The signals from and to the NC channel are not transferred to the PLC.
Bit = 1 The signals from the NC channel are transferred to the PLC (including M decoding M00 - M99 and output of the auxiliary functions and block information).

Notes:

- The bit can be set for the PLC.
- The bit is used if, for example, PLC 2 must monitor the signals of an NC channel; they are processed by PLC 1.
- With the SINUMERIK 880 GA2 a maximum of 8 (out of 16) channels can be activated.

Application examples:

Signals from/to the NC channel 1 to PLC 1: PLC MD 6000.0 = 1
Signals from the NC channel 1 to PLC 2: PLC MD 6103.0 = 1

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6006	Feed disable and read in disable on the NC channel							
DL 3	8	7	6	5	4	3	2	1
6007	Feed disable and read in disable on the NC channel							
DR 3	16	15	14	13	12	11	10	9

Bit = 1 Read in disable and feed disable are active.
Bit = 0 Read in disable and feed disable are inactive.

Notes:

- The bit is only active if the bit SIGNALS from the NC CHANNEL is set.
- In the PLC for which this bit is set the interface bit set for read in disable and feed disable are ignored on scanning the error and operational messages.
- The bit is only active in the modes AUTOMATIC and MDA.
- The mode selection switch must be transferred in the channel concerned.
- With the SINUMERIK 880 GA2 a maximum of 8 (out of 16) channels can be activated.

Application examples:

PLC 1 normally processes NC channel 1.
PLC 2 must initiate a read in disable **quickly** in special cases.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6009	M decoding with extended address for NC channel							
DR 4	8	7	6	5	4	3	2	1
6010	M decoding with extended address for NC channel							
DL 5	16	15	14	13	12	11	10	9

M DECODING WITH EXTENDED ADDRESS FOR NC CHANNEL (1 TO 16)

Bit = 0 M decoding with extended address is not performed.
Bit = 1 M decoding with extended address is performed.

Notes:

- To perform M decoding with extended address the corresponding data block (DB 80 - DB 95) must be loaded. If the bit is set and the DB is not available the PLC branches into the stop loop (for error list see Interface Description Part 1).
- The bit can be set for every PLC.
- The bit SIGNALS FROM/TO THE NC CHANNEL OR SIGNALS FROM THE NC CHANNEL must be set.
- With the SINUMERIK 880 GA2 a maximum of 8 (out of 16) channels can be activated.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6012	Signals from/to spindle							
DL 6			6	5	4	3	2	1

Bit = 0 Signals from/to spindle are not transferred to the spindle.
Bit = 1 Signals from/to spindle are transferred to the spindle.

Note:

The bit can only be set for one PLC as the signals to the spindle can only be output from one PLC. If the bits are set for example for PLC 1 and PLC 2 the PLC branches into the stop loop (for error list see Interface Description Part 1).

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6014	Signals from spindle							
DL 7			6	5	4	3	2	1

Bit = 0 Signals from the spindle are not transferred to the interface.
Bit = 1 Signals from the spindle are transferred to the interface.

Notes:

The bit be set for every PLC.

Application example:

Signals from/to spindle 2 are transferred to PLC 2: PLC MD 6114 bit 1=1
Signals from spindle 2 are transferred to PLC 1: PLC MD 6012 bit 1=1

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6016	Signals from/to axis							
DL 8	8	7	6	5	4	3	2	1
6017	Signals from/to axis							
DR 8	16	15	14	13	12	11	10	9
6018	Signals from/to axis							
DL 9	24	23	22	21	20	19	18	17

Bit = 0 Signals from/to axis are not transferred to the axis.
Bit = 1 Signals from/to axis are transferred to the axis by the PLC.

Note:

- The bit can only be set for one PLC as the signals to the axis can only be output by one PLC.
- If the bits for PLC 1 and PLC 2 are set the PLC branches into the stop loop (for error list see Interface Description Part 1).

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6021	Signals from axis							
DR 10	8	7	6	5	4	3	2	1
6022	Signals from axis							
DL 11	16	15	14	13	12	11	10	9
6023	Signals from axis							
DR 11	24	23	22	21	20	19	18	17

Bit = 0 Signals from the axis are not transferred to the interface.
Bit = 1 Signals from the axis are transferred to the interface.

Note:

The bit can be set for every PLC.

Application example:

Signals from/to axis 10 are transferred to PLC 2: PLC MD 6122.1=1
Signals from axis 10 are transferred to PLC 1: PLC MD 6022.1=1

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6026 **DL 13**	Serial interface		Deselec-tion auto-matic NC START DISABLE	Operator panel switchover active	Save FY 200-FY 225	Access by @ commands disabled	Command channel active	

Bit 7: Serial interface active

Note:

The bit can only be set for one PLC 1. If the bit is set for PLC 1 and PLC 2 the PLC branches into the stop loop (for error list see Interface Description part 1).

Bit 5: **Bit 5=0** The channel-specific signal NC START DISABLE (DB 10-25) (DW 11 bit 15) causes DISABLE NC START.
Bit 5=1 DISABLE NC START is not automatically initiated. It must be programmed by the user.

Bit 4: **Bit 4=0** Operator panel switchover is disabled.
Bit 4=1 Operator panel switchover is enabled.

Bit 3: **Bit 3=0** Save flags 224-255 with FB 65/66.
Bit 3=1 Save flags 200-255 with FB 65/66.

Bit 2: **Bit 2=0** The writing of PLC data is enabled by @ commands.
Bit 2=1 Writing is disabled.

Bit 1: **Bit 1=1** The function COMMAND CHANNEL is active.
Bit 1=0 The function COMMAND CHANNEL is not active.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6027 **DR 13**					Inputs from input submodule 4	3	2	1

Bit = 0 No transfer of input submodule signals into the process image of the inputs.
Bit = 1 Transfer of the input submodule signals into the process image of the inputs.

Assignment example:

Submodule (=rotary switch)	Input area
0	64 - 71
1	72 - 79
2	80 - 87
3	88 - 95

Notes:

- The bit can be set for every PLC.
- Up to 4 I/O submodules can be connected to the NC operator panels. The input signals of the submodules can be routed to the process image of PLC 1 to PLC 2 via the PLC machine data ACCESS TO I/O SUBMODULE 1 TO 4.
- Maximum value: 96.
  The 1st input byte for operator panel inputs can be set via PLC MD 6.

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6028 **DL 14**					Outputs to output submodule 4	3	2	1

Bit = 0 No transfer to the output module from the process image of the outputs.
Bit = 1 Transfer to the output module from the process image of the outputs.

Assignment example:

Submodule (=rotary switch)	Output area
0	64 - 67
1	68 - 71
2	72 - 75
3	76 - 79

Notes:

- The bit can be set for one PLC as the output from the outputs can only come from one PLC. If the bits for PLC 1 and PLC 2 are set the PLCs branch into the stop loop (for error list see Interface Description Part 1).
- Up to 4 I/O submodules can be connected to the operator panel. The output signals of the submodules can be routed to the PLC via PLC machine data ACCESS TO I/O SUBMODULE 1 TO 4.
- Maximum value=112.
  The 1st output byte for operator panel outputs can be set via PLC MD 7.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6029 **DL 6**					Message in target channel	ELG signals to NC		T/H word jumpering

Bit 0 = 1 The T/H words are transferred to the expanded interface, (see Interface Description Part 1, Signals, Section 6, for explanation).
= 0 No transfer to the expanded interface.
Bit 2 = 1 The transfer of the ELG-specific interface signals (DB 29) from the PLC to the NC must be enabled by setting a machine bit (PLC MD 6029 bit 2 for PLC 1, PLC MD 6129 bit 2 for PLC 2). The bit can be set for only one PLC.
Bit 3 = 1 Jumpering suppressed" message (DB 10-25, DW 63 bit 6) appears in the source channel (standard).
Bit 3 = 0 Jumpering suppressed" message appears in the target channel.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6030 **DL 15**	Error/operational messages for not activated channels							
	8	7	6	5	4	3	2	1
6031 **DR 15**	16	15	14	13	12	11	10	9

0 signal: Corresponding **inactive** channel DB is not used for the activation of error and operational messages.
1 signal: The inactive channel DB is used for the activation of error and operational messages.

Example:

0	0	0	0	0	1	1	1

PLC MD 6000

Only the error bits of the active channel DBs are evaluated for channel 1 to 3.

0	0	0	1	1	0	0	0

PLC MD 6030

In addition, the error bits of the inactive channel DBs of channels 4 and 5 are evaluated.

Note:

If an error bit is set in an inactive channel DB which is used for the extended display of error and operational messages, then the corresponding errors do **not** appear for the display of message groups.
The simutaneous assignment of channels to the PLC (MD 6*00 and 6*01) and the activation of the display of error and operational messages of channels **not** used (MD 6*30 and 6*31) to different PLCs is not permissible because it causes incorrect displays.

Example: MD 6000.0 = 1 and
MD 6030.0 = 1

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6032	Error messages signals to NC channel							
DL 16	DR 9	DL 9	DR 8	DL 8	DR 7	DL 7	DR 6	DL 6
6033	Error messages signals to NC channel							
DR 16					DR 11	DL 11	DR 10	DL 10

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be set for PLC 1 and PLC 2
- The bit applies to all channels.
- If for the same PLC the **same** bit is also set for operational messages, the PLC branches into the stop loop in the start-up routine.

Application example:

Error messages for 1 signal of a bit in DR 8 from: PLC MD 6032.
DR 8 bit 5 in channel 2 (DB 11) causes error message No. 6145 (see Section 19 of the Interface Description).

Activating error and operational messages via unused channel DBs.

0 signal: Corresponding **inactive** channel DB is not used for the activation of error and operational messages.

1 signal: The inactive channel DB is used for the activation of error and operational messages.

Note:

If an error bit is set in an unused channel DB which is used for the extended display of error and operational messages, the corresponding errors do not appear when message groups are displayed.

Example:

0	0	0	0	0	1	1	1	PLC MD 6000

Only the error bits of the active channel DBs are evaluated for channel 1 to 3.

0	0	0	1	1	0	0	0	PLC MD 6030

The error bits of the inactive channel DBs of channels 4 and 5 are evaluated.

Table of message numbers

Channel 1	6000 - 6095	DB 10, DW 6 - DW 11
Channel 2	6100 - 6195	DB 11, DW 6 - DW 11
Channel 3	6200 - 6295	DB 12, DW 6 - DW 11
Channel 4	6300 - 6395	DB 13, DW 6 - DW 11
Channel 5	6400 - 6495	DB 14, DW 6 - DW 11
Channel 6	6500 - 6595	DB 15, DW 6 - DW 11
Channel 7	6600 - 6695	DB 16, DW 6 - DW 11
Channel 8	6700 - 6795	DB 17, DW 6 - DW 11
Channel 9	6800 - 6895	DB 18, DW 6 - DW 11
Channel 10	6900 - 6995	DB 19, DW 6 - DW 11
Channel 11	7000 - 7095	DB 20, DW 6 - DW 11
Channel 12	7100 - 6715	DB 21, DW 6 - DW 11
Channel 13	7200 - 7295	DB 22, DW 6 - DW 11
Channel 14	7300 - 7395	DB 23, DW 6 - DW 11
Channel 15	7400 - 7495	DB 24, DW 6 - DW 11
Channel 16	7500 - 7595	DB 25, DW 6 - DW 11
Spindle 1	8000 - 8015	DB 31, DW 3
Spindle 2	8020 - 8035	DB 31, DW 7
Spindle 3	8040 - 8055	DB 31, DW 11
Spindle 4	8060 - 8075	DB 31, DW 15
Spindle 5	8080 - 8095	DB 31, DW 19
Spindle 6	8100 - 8115	DB 31, DW 23
Axis 1	8200 - 8215	DB 32, DW 3
Axis 2	8220 - 8235	DB 32, DW 7
Axis 3	8240 - 8255	DB 32, DW 11
Axis 4	8260 - 8275	DB 32, DW 15
Axis 5	8280 - 8295	DB 32, DW 19
Axis 6	8300 - 8315	DB 32, DW 23
Axis 7	8320 - 8335	DB 32, DW 27
Axis 8	8340 - 8355	DB 32, DW 31
Axis 9	8360 - 8375	DB 32, DW 35
Axis 10	8380 - 8395	DB 32, DW 39
Axis 11	8400 - 8415	DB 32, DW 43
Axis 12	8420 - 8435	DB 32, DW 47
Axis 13	8440 - 8455	DB 32, DW 51
Axis 14	8460 - 8475	DB 32, DW 55
Axis 15	8480 - 8495	DB 32, DW 59
Axis 16	8500 - 8515	DB 32, DW 63
Axis 17	8520 - 8535	DB 32, DW 67
Axis 18	8540 - 8555	DB 32, DW 71
Axis 19	8560 - 8575	DB 32, DW 75
Axis 20	8580 - 8595	DB 32, DW 79
Axis 21	8600 - 8615	DB 32, DW 83
Axis 22	8620 - 8635	DB 32, DW 87
Axis 23	8640 - 8655	DB 32, DW 91
Axis 24	8660 - 8675	DB 32, DW 95
PLC 1 PLC 2	9000 - 9997	DB 58, DW 1 - DL 32 DB 58, DW 1 - DL 32

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6034	Error messages signals to spindle							
DL 17							DR $_{k+3}$	DL $_{k+3}$

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.

Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- The bit applies to **all** spindles.
- If for the same PLC the **same** bit is also set for operational messages, the PLC branches into the stop loop in the start-up routine (for error list see Interface Description Part 1).

Application example:

DB 31 D 7.9 1 signal error message 8021;
PLC MD 6034.0= 1 (see Section 10 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6035	Error messages signals to axis							
DR 17							DR $_{k+3}$	DL $_{k+3}$

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.

Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- The bit applies to **all** axes.
- If for the **same** PLC the same bit is also set for operational messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 32 D 3.3 1 signal error message 8211;
PLC MD 6035 Bit 1= 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6036	Error messages DB 58 messages							
DL 18	DR 4	DL 4	DR 3	DL 3	DR 2	DL 2	DR 1	DL 1
6037	Error messages DB 58 messages							
DR 18	DR 8	DL 8	DR 7	DL 7	DR 6	DL 6	DR 5	DL 5
6038	Error messages DB 58 messages							
DL 19	DR 12	DL 12	DR 11	DL 11	DR 10	DL 10	DR 9	DL 9
6039	Error messages DB 58 messages							
DR 19		DL 16	DR 15	DL 15	DR 14	DL 14	DR 13	DL 13

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- If for the **same** PLC the same bit is also set for operational messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 58 (PLC 2) D 3.10 1 signal error message 9285 ;
PLC MD 6136.4 = 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6040	Operational messages signals to NC channel							
DL 20	DR 9	DL 9	DR 8	DL 8	DR7	DL 7	DR 6	DL 6
6041	Operational messages signals to NC channel							
DR 20					DR 11	DL 11	DR 10	DL 10

Bit = 0 The bits of the corresponding interface byte are not evaluated for operational messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for operational messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- The bit applies to **all** NC channels.
- If for the **same** PLC the same bit is also set for error messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 10 D 0.6 1 signal operational message 6000;
PLC MD 6040.0= 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6042	Operational messages signal to spindle							
DL 21							DR $_{k+3}$	DL $_{k+3}$

Bit = 0 The bits of the corresponding interface byte are not evaluated for operational messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- The bit applies to **all** spindles.
- If for the **same** PLC the same bit is also set for error messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 31 D 7.1 1 signal operational message 8029;
PLC MD 6042.1= 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6043	Operational messages signals to axis							
DR 21							DR $_{k+3}$	DL $_{k+3}$

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for operational messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- The bit applies to **all** axes.
- If for the **same** PLC the same bit is also set for error messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 32 D 3.15 1 signal operational message 8207;
PLC MD 6043.0= 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6044	Operational message DB 58 messages							
DL 22	DR 4	DL 4	DR 3	DL 3	DR 2	DL 2	DR 1	DL 1
6045	Operational message DB 58 messages							
DR 22	DR 8	DL 8	DR 7	DL 7	DR 6	DL 6	DR 5	DL 5
6046	Operational message DB 58 messages							
DL 23	DR 12	DL 12	DR 11	DL 11	DR 10	DL 10	DR 9	DL 9
6047	Operational message DB 58 messages							
DR 23		DL 16	DR 15	DL 15	DR 14	DL 14	DR 13	DL 13

Bit = 0 The bits of the corresponding interface byte are not evaluated for operational messages by the PLC operating system.

Bit = 1 The bits of the corresponding interface byte are evaluated for operational messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- If for the **same** PLC the same bit is also set for error messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 58 (PLC I) D 5.1 1 signal operational message 9073;
PLC MD 6045.1 = 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6048 **DL 24**	Processing delay of OB 7	OB 6	OB 5	OB 4	OB 3	OB 2		

Bit = 0 A processing delay in the corresponding OB does not cause a STOP in the AG.
Bit = 1 A processing delay in the corresponding OB does cause a STOP in the AG.

Note:

If a stop in the AG is not required for a processing delay in the OB, a bit in flag byte 6 is set when the processing delay occurs. By scanning this bit the user can take any action he wishes.

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6049 **DR 24**							IP/WF modules inserted	Access to PLC I/Os

Bit 1 = 0 No IP or WF module inserted in the expansion unit (EU 185) (standard).
Bit 1 = 1 IP/WF modules inserted; a cold restart is enforced after each RESET.

With Dual PLCs, both MD bits must be the same (same power-up routine for both PLCs).

Bit 0 = 1: Fast acknowledgement (no exact timeout analysis on error)
Bit 0 = 0: Slow acknowledgement (exact timeout analysis on error)

Note:

Set bit 0=0 on startup or test.
Set bit 0=1 during operation.

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
6050 **DL 25**	Disable of OB 7	OB 6	OB 5	OB 4	OB 3	OB 2		

Bit = 0 The corresponding OB is enabled for processing.
Bit = 1 The OB is disabled for processing and is not called by the system program.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6051 **DR 25**							PG mode	PLC mode

Bit 1= 0 : The PLC 135 WB signals itself to be a 155U to the programmer.
S5-DOS version (155 U) restricted functionality.

Bit 1= 1: S5-DOS version + and additional software (PLC 135 WB)
or S5-DOS version
With this software configuration +the PLC 135 WB signals itself with its own identifiers (ISTACK etc.) to the programmer.

Bit 0= 1: A change of processing level (e.g. OB 5 interrupts OB 6) is only possible at block boundaries (behaviour like PLC 113 WB normal mode).

Bit 0= 0: A change of processing level is possible after every STEP 5 command (special mode).

Note:

The bit PLC mode must = 0 when the OB 2 and OB 5 are called. If this bit is set and the OBs are loaded in the AG they are still not processed.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6060 **DL 30**					Stop on failure of Servo CPU 4	Servo CPU 3	Servo CPU 2	Servo CPU 1
6061 **DR 30**					Stop on failure of		NC CPU 2	NC CPU 1

Bit 3-1 = 0 : If the CPU in question fails the PLC branches into the stop loop.
Bit 3-1 = 1: No stop

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6063 **DR 31**							Always 1	

Bit1: This bit must always be set to "1".

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6068 **DL 34**	Edge selection of interrupt inputs of 1st interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6069 **DR 34**	Edge selection of interrupt inputs of 1st interface DMP module 2nd line							
	7	6	5	4	3	2	1	0
6070 **DL 35**	Edge selection of interrupt inputs of 2nd interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6071 **DR 35**	Edge selection of interrupt inputs of 2nd interface DMP module 2nd line							
	7	6	5	4	3	2	1	0
6072 **DL 36**	Edge selection of interrupt inputs of 3rd interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6073 **DR 36**	Edge selection of interrupt inputs of 3rd interface DMP module 2nd line							
	7	6	5	4	3	2	1	0

The interrupt is triggered when a signal change occurs at the interrupt input. Operating system machine data bits can be used for each individual interrupt input to specify whether the positive or negative edge triggers the interrupt.

Input values: 0: Positive edge triggers interrupt
1: Negative edge triggers interrupt

Standard value: 0: The positive edge triggers the interrupt at all interrupt inputs that are enabled.

The system program checks the permissibility of the machine data for the interrupt triggering I/Os on cold restarts. If impermissible machine data are identified, the program outputs an error message and branches to the stop state.

Notes:

- The interrupt bytes are selected with PLC MDs 130 to 135.
- The interrupt inputs must be enabled by PLC MDs 6074 to 6079.
- These PLC MDs are ineffective with "free configuration" (see PLC MD 136).

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6074 DL 37	Enable interrupt inputs of 1st interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6075 DR 37	Enable interrupt inputs of 1st interface DMP module 2nd line							
	7	6	5	4	3	2	1	0
6076 DL 38	Enable interrupt inputs of 2nd interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6077 DR 38	Enable interrupt inputs of 2nd interface DMP module 2nd line							
	7	6	5	4	3	2	1	0
6078 DL 39	Enable interrupt inputs of 3rd interface DMP module 1st line							
	7	6	5	4	3	2	1	0
6079 DR 39	Enable interrupt inputs of 3rd interface DMP module 2nd line							
	7	6	5	4	3	2	1	0

Each individual interrupt input can be disabled (0) or enabled (1). The interrupt inputs are entered in FYs 8 to 11 when the signal state changes. The choice of edge is made with PLC MDs 6068 to 6079.

Notes:

- These PLC MDs are ineffective with "free configuration" (see PLC MD 136).
- The interrupt bytes are selected with PLC MDs 130 to 135.

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6080	Error messages DB 58 messages							
DL 40	DR 20	DL 20	DR 19	DL 19	DR 18	DL 18	DR 17	DL 17
6081	Error messages DB 58 messages							
DR 40	DR 24	DL 24	DR 23	DL 23	DR 22	DL 22	DR 21	DL 21
6082	Error messages DB 58 messages							
DL 41	DR 28	DL 28	DR 27	DL 27	DR 26	DL 26	DR 25	DL 25
6083	Error messages DB 58 messages							
DR 41		DL 32	DR 31	DL 31	DR 30	DL 30	DR 29	DL 29

Bit = 0 The bits of the corresponding interface byte are not evaluated for error messages by the PLC operating system.
Bit = 1 The bits of the corresponding interface byte are evaluated for error messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- If for the **same** PLC the same bit is also set for operational messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 58 (PLC 2) D 3.10 1 signal error message 9285 ;
PLC MD 6136.4 = 1 (see Section 19 of the Interface Description)

PLC MD DW No.	Bit No. 7	6	5	4	3	2	1	0
6084	Operational messages DB 58 messages							
DL 42	DR 20	DL 20	DR 19	DL 19	DR 18	DL 18	DR 17	DL 17
6085	Operational messages DB 58 messages							
DR 42	DR 24	DL 24	DR 23	DL 23	DR 22	DL 22	DR 21	DL 21
6086	Operational messages DB 58 messages							
DL 43	DR 28	DL 28	DR 27	DL 27	DR 26	DL 26	DR 25	DL 25
6087	Operational messages DB 58 messages							
DR 43		DL 32	DR 31	DL 31	DR 30	DL 30	DR 29	DL 29

Bit = 0 The bits of the corresponding interface byte are not evaluated for operational messages by the PLC operating system.

Bit = 1 The bits of the corresponding interface byte are evaluated for operational messages by the PLC operating system.

Notes:

- The bit can be different in PLC 1 and PLC 2.
- If for the **same** PLC the same bit is also set for error messages, the PLC branches into the stop loop (for error list see Interface Description Part 1).

Application example:

DB 58 (PLC I) D 5.1 1 signal operational message 9073;
PLC MD 6045.1 = 1 (see Section 19 of the Interface Description)

9.4.3.2 General PLC MD bits

PLC machine data 6400 to 6519 are omitted on the SINUMERIK 880 GA2 because it does not process any global I/O devices.

PLC machine data 6560 to 6583 (PLC STOP with I/O module errors) are omitted on the SINUMERIK 880 GA2. The PLC always enter the STOP state in the event of such errors.

9.4.3.3 PLC MD bits for function blocks (DB 64)

PLC MD	Bit No.							
	7	6	5	4	3	2	1	0
7000 to 7249								
7250 to 7499								

PLC 1	PLC 2
PLC MD No. 7000 to 7249	PLC MD No. 7250 to 7499

Please refer to the relevant FB Package Description for an explanation of the PLC MD bits.

FB Package 0: For explanation see Interface Description Part 1, Signals, Section 19: Display programs for PLC data and messages;
FB Package 1 + 2: Tool management;
FB Package 4 + 5: Computer link.

9.4.3.4 PLC MD bits for user (DB 65)

PLC MD DW No.	Bit No.							
	7	6	5	4	3	2	1	0
8000 to 8049			6	5	4	3	2	1
8050 to 8099								

PLC 1	PLC 2
PLC MD No. 8000 to 8049	PLC MD No. 8050 to 8099

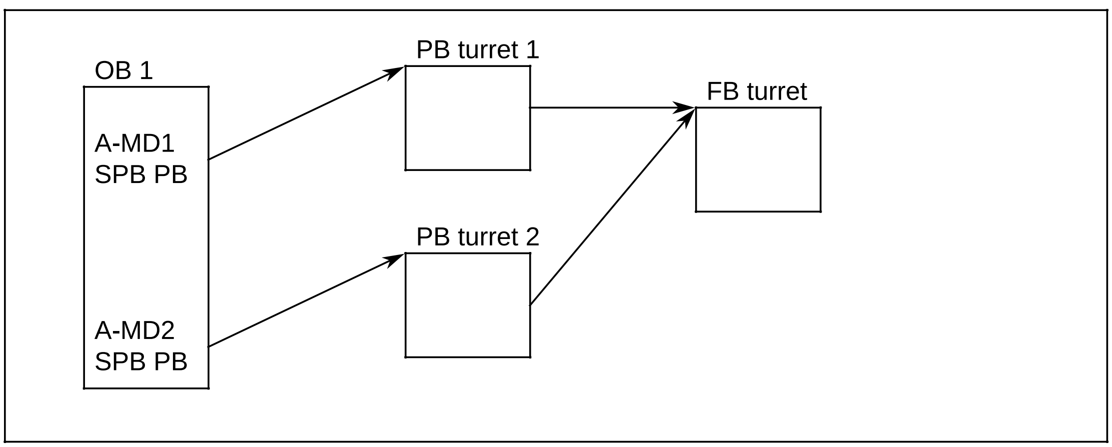
In addition to the PLC MD words, the PLC MD bits are also availble to the user. This bit area covers 25 words (400 bits).

The machine manufacturer can process program blocks, function blocks or parts of his program depending on the bits set and assign machine-specific option bits with the PLC machine data. The user can also assign machine-specific values using these values.

Application example:

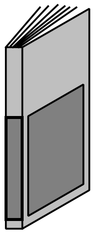
A user constructs machines with different turrets. But the user would like to supply the same program for all machines. The turrets differ in the number of tool locations, e.g. turret 1 has 6 tool locations, turret 2 has 8 locations.

The program for every type of turret (here program block) is called via the corresponding PLC machine data bit. The number of locations is predefined in the program via the PLC MD words.



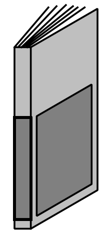
9.5 Cycle machine data (Cycle MD)

The cycle machine data are active only if you use the measuring cycles from Version 20 onwards. A detailed description of the cycle machine data is contained in:

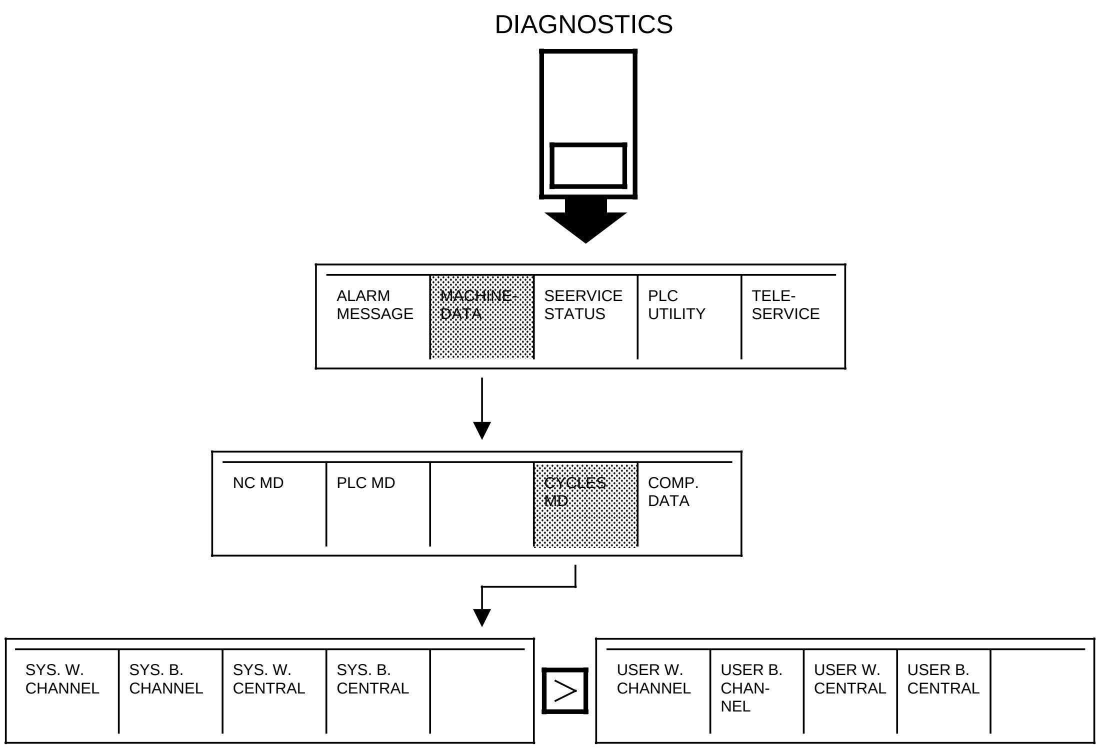


Start-up Guide
Measuring Cycles, Version 20 and 30

(Available from: LZW-Lager, Fürth Bislohe
Order No.: see List of SINUMERIK Documentation)



When working with subroutines and cycles, not only are variable values (R parameters) required for calculations but also fixed values or values which are stored in safe memories. For this purpose, from Software Version 3, cycle MD and cycle SD were introduced
(see Section 9.7).



SYS. W.	System words
SYS. B.	System bits
USER W.	User words
USER B.	User bits
CHANNEL	Channel-specific
CENTRAL	Central (for all channels together)

The cycle MDs are divided into two areas as follows (systems areas Siemens and user area).

Channel-specific cycle MDs:

0 . 49	System words (Siemens)
400 . 449	User words
800 . 849	System bits (Siemens)
900 . 949	User bits

Central cycle MDs:

1000 . 1149	System words (Siemens)
4000 . 4149	User words
7000 . 7049	System bits (Siemens)
8000 . 8049	User bits

The channel-specific cycle MDs are deleted in installation mode with the softkey "clear cycle MD". A standard setting for cycle MDs does not exist. Before input of cycle MDs on the NC operator panel the password .... must be entered.

The system cycle MDs (words and bits) are reserved exclusively by Siemens (Siemens Nuremberg). A description of the assignment can be found with the appropriate subroutines or cycles, with which the cycle MDs are used (e.g. measuring cycles version 20).

The user cycle MDs (words and bits) are for the manufacturer (or for the final user) and are therefore not assigned by Siemens (only for manufacturer-specific applications at the request of the manufacturer).

The cycle MDs can only be read and written with the following CL800 commands (see also Programming Guide):

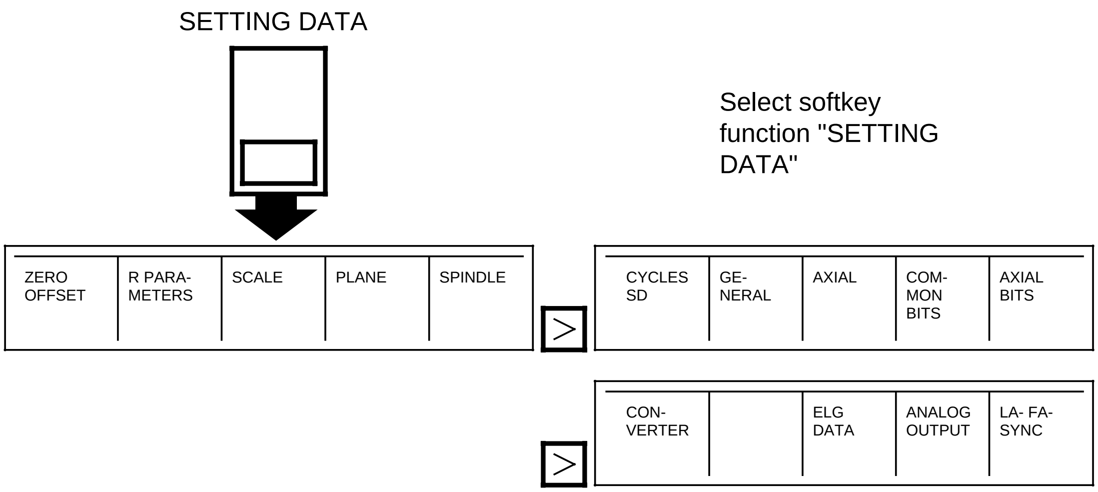
@ 303	Read cycle MD	Word
@ 304	—— " ——	Byte
@ 305	—— " ——	Bit
@ 403	Write cycle MD	Word
@ 404	—— " ——	Byte
@ 405	—— " ——	Bit

The cycle MDs can be read by the PLC with FB 61 and written by the PLC with FB 62.

9.6 NC setting data (SD)

9.6.1 General

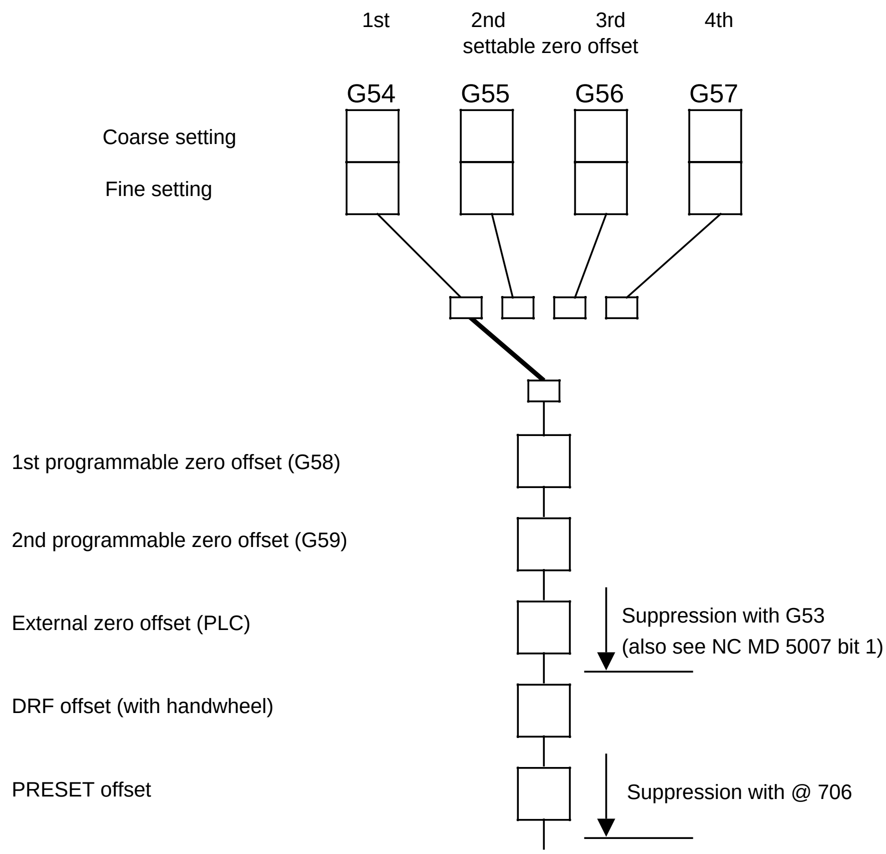
All setting data (SD) are active immediately (without "POWER ON"). If program execution is taking place, they are active in the next block if they were modified with G functions. The general SD bits and the SD bits for the serial interfaces can also be modified in the "OVERALL RESET" installation mode:



Select softkey function "SETTING DATA"

9.6.2 Zero offsets (ZO)

G54	1st	settable zero offset	(coarse + fine)
G55	2nd	settable zero offset	(coarse + fine)
G56	3rd	settable zero offset	(coarse + fine)
G57	4th	settable zero offset	(coarse + fine)

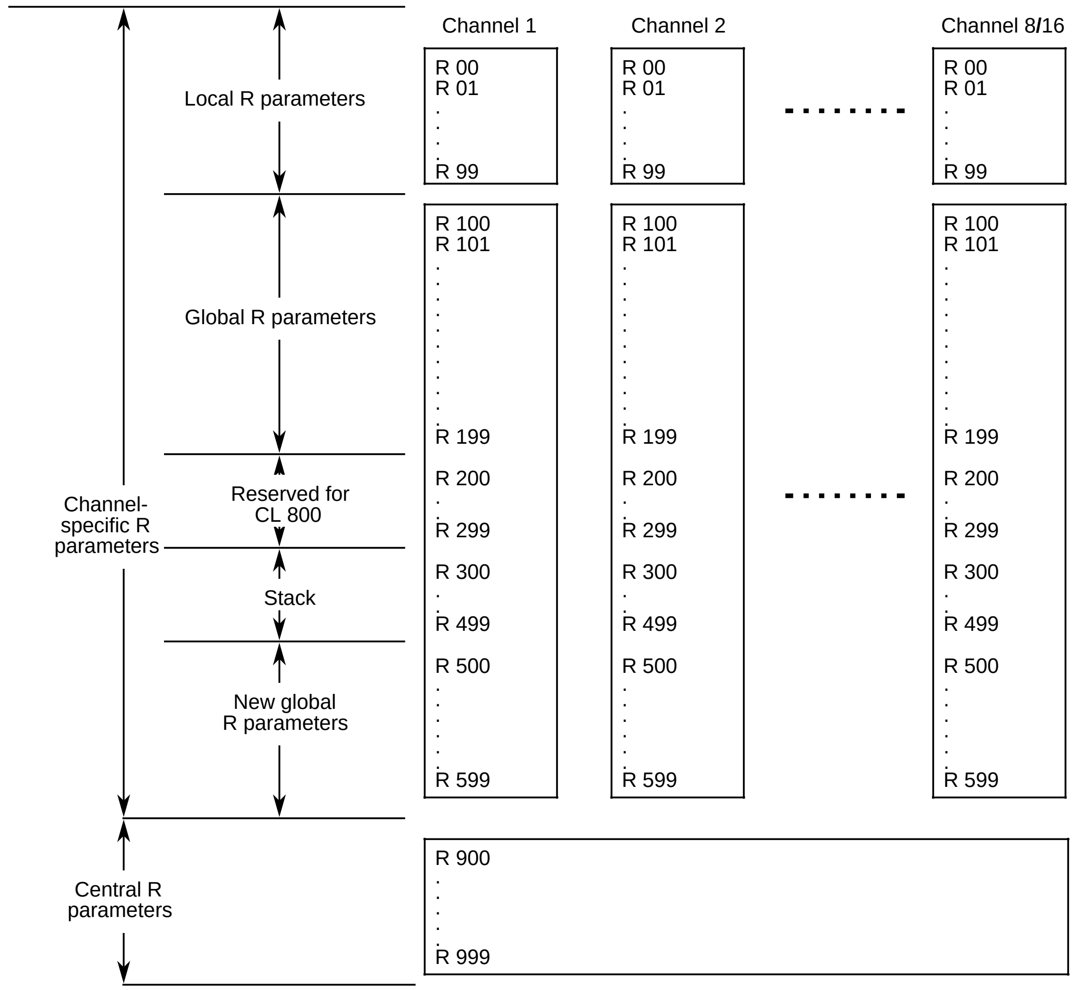


The **initial setting** for programming is **G54** !

G58	1st programmable ZO
G59	2nd programmable ZO
External ZO	External ZO from PLC The values are transmitted from the PLC via external data input. They may only be deleted with the PLC or using the "USER MEMORY FORMAT" softkey.

9.6.3 R parameters

The overall system has 600 parameters per channel and 100 central parameters:
Parameters R0 to R599 are channel-specific,
Parameters R900 to R999 are valid for all channels.



R parameter assignment

R0 – R49: **Typical application per channel:**

Input of cycles and subroutines.

R50 – R99: **Typical application per channel:**

For calculations within cycles and subroutines. The same local parameters may be used for nested subroutines. When cycles or subroutines are called with @ 040 ... 043, an R parameter stack saves the data used so far and stores them after return to the calling program.

R100 –	R199:	**Typical application per channel:** Memory for data which must be accessible for the main programs and subroutines. R100 - R109 are assigned if Siemens tool management is used. R110-R199 are free for the user.
R200 –	R219:	Siemens assignment (cycle converter)
R220 –	R239:	CL 800 compiler on WS800 compiler (Siemens assignment)
R240 –	R299:	Still unassigned; assigned internally by Siemens as required !
	R300:	Stack pointer for @ 040, @ 041, @ 042, @ 043 (set to 301 with each M2, M30 RESET)
R301 –	R499:	Stack area for @ 040, @ 041, @ 042, @ 043
R500 –	R599:	100 new global R parameters per channel Use: as for R100 - R199
R900 –	R999:	**Typical application:** Higher-level memory for all NC channels, e.g. for buffering target positions used by another channel.

Note:

R parameter areas R100 to R199, R500 to R599 and R900 to R999 are provided for the user as standard.

The user can also use the other R parameters if he is not using the associated functions (CL 800 language, SIEMENS cycles or similar).

9.6.4 General setting data

0	Dry run feedrate		0
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**1072 0000**	**1000 units/min**

If "dry run feedrate" is selected on the control, the tool path feedrate selected is the dry run feedrate (mm/min (G94)) as opposed to the programmed feedrate.

1	Dyn. smoothing time thread cutting		1
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**5**	**2^n-1 ×IPO cycle**

If the feedrate depends on the spindle speed (e.g. in the case of thread cutting), the NC determines the spindle speed fluctuation in the position control cycle and modifies the feedrate with an appropriate setpoint output.

If a value greater than 1 is entered here, the spindle speed fluctuations are averaged over the stated IPO cycles before being used to modify the feedrate.

The smoothing time constant can also be modified by programming G 92 T ... .

Input value	0	1	2	3	4	5
IP cycle times	0	1	3	7	15	31
Setpoint output feedrate	Jump		Ramp			

9.6.5 ELG setting data values

Setting data No. FA1	FA2	FA3	Speed ratio parameter	LA	$K_ü$
SD10	**SD20**	**SD30**	Numerator	1	1st fictitious leading axis
SD11	**SD21**	**SD31**	Denominator		
SD12	**SD22**	**SD32**	Numerator	2	2nd fictitious leading axis
SD13	**SD23**	**SD33**	Denominator		
SD14	**SD24**	**SD34**	Numerator	3	1st real leading axis
SD15	**SD25**	**SD35**	Denominator		
SD16	**SD26**	**SD36**	Numerator	4	2nd real leading axis
SD17	**SD27**	**SD37**	Denominator		
SD18	**SD28**	**SD38**	Numerator	5	3rd real leading axis
SD19	**SD29**	**SD39**	Denominator		

Input unit: Floating-point format with sign
Max. input value: 99 999 999 (8 digits)
- Max. 7 places after the decimal point; e.g. 0.1234567

Max. $K_Ü$ 1000

Explanation: See Function Manual "Electronic Gearbox"

9.6.6 Setting data for analog outputs

SD No.	Analog channel	Designation	Standard value	Maximum input value	Input unit
40	1	Assignment of output cell	0	179	–
41		Factor for right shift or division	0	255	1
42		Fact. for left shift or multiplic.	0	– 127 to + 128	1
43		Offset value	0	± 16383	1
44	2	Assignment of output cell	0	179	–
45		Factor for right shift or division	0	255	1
46		Fact. for left shift or multiplic.	0	– 127 to + 128	1
47		Offset value	0	± 16383	1
48	3	Assignment of output cell	0	179	–
49		Factor for right shift or division	0	255	1
50		Fact. for left shift or multiplic.	0	– 127 to + 128	1
51		Offset value	0	± 16383	1
52	4	Assignment of output cell	0	179	1
53		Factor for right shift or division	0	255	1
54		Fact. for left shift or multiplic.	0	– 127 to + 128	1
55		Offset value	0	± 16383	1

Explanation: See Function Manual "Electronic Gearbox"

Note:

Setting data 40 to 43 can also be used for starting up the "Tapping without compensating chuck" function. See Section 11.15 for an explanation.

9.6.7 Setting data values for synchronization of following axis (FA) with leading axis (LA)

SD No.	Designation	Standard value	Maximum input value	Reference system	Input unit
56	Synchronization position FA1	0	1 070 000	IS	1)
57	Synchronization position of LA for FA1	0	1 070 000	IS	1)
58	LA-Nummer für Synchronisation mit FA1	0	5	–	–
59	Synchronization position FA2	0	1 070 000	IS	1)
60	Synchronization position of LA for FA2	0	1 070 000	IS	1)
61	LA-Nummer für Synchronisation mit FA2	0	5	–	-
62	Synchronization position FA3	0	1 070 000	IS	1)
63	Synchronization position of LA for FA3	0	1 070 000	IS	1)
64	LA number for synchronization with FA3	0	5	–	–

Explanation: See Function Manual "Electronic Gearbox".

1) *Number of decimal places depends on input resolution*

9.6.8 Setting data values for synchronous spindles

SD No.	Change	Designation	Standard value	Maximum input value	Input unit
14		Speed ratio FS 1, Numerator	0	99 999 999	–
15		Speed ratio FS 1, Denominator	1	99 999 999	–
24		Speed ratio FS 2, Numerator	0	99 999 999	–
25		Speed ratio FS 2, Denominator	1	99 999 999	–
34		Speed ratio FS 3, Numerator	0	99 999 999	–
35		Speed ratio FS 3, Denominator	1	99 999 999	–

Explanation: See Function Manual "Extended Spindle Functions".

9.6.9 Channel-specific setting data

200*	Scale factor		200*
Standard value	Lower input limit	Upper input limit	Units
1	**0.00001**	**99.99999**	**–**

If the scale factor is selected with G51, all subsequently programmed axis values are modified with the scale factor (see Programming Guide for exact description).

The scale factor is programmed together with G51 in the block.

204*	Thread starting angle		204*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**359.999**	**degrees**

A multiple thread can be produced very easily by stating a thread starting angle for thread cutting. This setting data can be modified at the user interface or by programming "G92 A ..." in the part program. To allow this, however, the "Extended thread package" Option must be active.

The angle entered last is retained on Power Off.

9.6.10 Axial setting data values

300*	Minimum working area limitation		300*
Standard value	Lower input limit	Upper input limit	Units
– 99999.999	**– 99999.999**	**+ 99999.999**	**mm or inch**

304*	Maximum working area limitation		304*
Standard value	Lower input limit	Upper input limit	Units
+ 99999.999	**– 99999.999**	**+ 99999.999**	**mm or inch**

Working area limitation makes it possible to restrict the traversing ranges in the automatic and/or JOG mode (additional to software limit switches). Axis display is mode group-specific. The working area limitations can be modified in the program with G25/26.

The working area limitations do not have any effect in the "reference point approach" mode.

312*	Scale centre NC		312*
Standard value	Lower input limit	Upper input limit	Units
0	**– 99999.999**	**+ 99999.999**	**mm or inch**

The scale centre defines the location of the reference point for modifying the programmed axis positions by the scale factor (see Programming Guide for exact description).

The scale centre is programmed together with G51 in the block.

316*	Scale centre simulation		316*
Standard value	Lower input limit	Upper input limit	Units
0	**– 99999.999**	**+ 99999.999**	**mm or inch**

To allow graphic simulation parallel to processing, the programmed scale centre for simulation is entered in another NC SD (no coincidence of data with the processing NC).

9.6.11 Spindle-specific setting data

401*	Programmable spindle speed limit for G96		401*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**99 999**	**rev/min or 0.1 rev/min**

The programmed limit for spindle speed limits the speed of the spindle at constant cutting rate (G96). The setting data can be altered in the program by means of command G92.

Units

The units of the value entered depend on MD 520*, bit 3:

Bit 3 = 0: Unit rev/min
Bit 3 = 1: Unit 0.1 rev/min

Note

The maximum value of spindle speed is determined by the minima of the following values:

- MD 403* to 410* "Max. speed per gear stage"
- MD 451* "Max. chuck speed"
- SD 401* "Programmable spindle speed limit for G96"; programmed with G26
- SD 403* "Programmable spindle speed limit"; programmed with G26

402*	Spindle position for M19		402*
Standard value	Lower input limit	Upper input limit	Units
0	**0**	**35 999**	**0.01°**

This setting data establishes the angle to which the spindle is positioned when M19 without an S value is programmed in the part program (or MDA, overstore).

If an S value is programmed with M19 it will be accepted into the setting data.

Examples

- M19 S270 L_F positions the spindle to 270 degrees. The angle is entered in the setting data.
- M19 L_F positions the spindle to whatever angle has been entered in the setting data.

403*	Programmable spindle speed limit		403*
Standard value	Lower input limit	Upper input limit	Units
100	**0**	**99 999**	**rev/min or 0.1 rev/min**

The maximum speed of the spindle is limited to this value. The setting data can be altered in the program by means of command G26.

Units

The units of the values entered depend on MD 520*, bit 3:

Bit 3 = 0:	Unit	rev/min
Bit 3 = 1:	Unit	0.1 rev/min

Note

The maximum value of the spindle speed is determined by the minima of the following values:

- MD 403* to 410* "Max. speed per gear stage"
- MD 451* "Max. chuck speed"
- SD 401* "Programmable spindle speed limit for G96"; programmed with G92
- SD 403* "Programmable spindle speed limit"; programmed with G26

9.6.12 General NC setting data

5000		SD No. / Bit No.				5000	
7	6	5	4	3	2	1	0
					Funct. expansion of UMS03/04 usable		
Calculation of overrun compen-sation					L95/L93	Drilling patterns L903/L930	L81 - 89 and L98

Bit 2,1, 0: Effective with user memory submodule (UMS) version 03/04
Bit 2, 1, 0 = 0 Compatible with UMS 02

Bit 7: Calculation of overrun compensation for L84 is activated. See Cycles Documentation UMS 04 for further information.

5001		SD No. / Bit No.				5001	
7	6	5	4	3	2	1	0
Addition Channel 3 to Channel 4	Addition Channel 1 to Channel 2						Display workpiece-oriented actual

Bit 0: The actual value display for axes (actual position) refers to the workpiece zero point and not to the machine zero point (reference point).

Bit 6, 7: See Function Manual "Electronic Gearbox".

5003		SD No. / Bit No.				5003	
7	6	5	4	3	2	1	0
Configuration of analog channels							
Analog channel 4		Analog channel 3		Analog channel 2		Analog channel 1	
Mul/Div	ON	Mul/Div	ON	Mul/Div	ON	Mul/Div	ON

Explanation: See Function Manual "Electronic Gearbox"

9.6.13 ELG setting data bits

SD No.	Bit No.							
	7	6	5	4	3	2	1	0
FA1: 5004 FA2: 5006 FA3: 5008	Calculate new $K_ü$	Acceler-ation limit-ation syn-chronous	Enable emergency retraction	Com-pensatory controller ON		FA overlay ON	Syn-chroniz-ation ON	LINK ON
FA1: 5005 FA2: 5007 FA3: 5009	Activate new $K_ü$		Synchro-nization START					

Explanation: See Function Manual "Electronic Gearbox".

9.6.14 Synchronous spindle setting data bits

SD No.	Bit No.							
	7	**6**	**5**	**4**	**3**	**2**	**1**	**0**
FS1 5004 FS2 5006 FS3 5008	Calculate new $K_ü$	Acceler-ation limit-ation syn-chronous	Enable emergen-cy retraction	Com-pensatory controller ON		FS overlay ON		FS in synchro-nous mode
FS1 5005 FS2 5007 FS3 5009	Activate new $K_ü$							

Explanation: See Function Manual "Extended Spindle Functions".

9.6.15 NC setting data bits for serial interfaces

(See Planning Guide Universal Interface 800 for a detailed description)

SD No.	Bit No. 7	6	5	4	3	2	1	0
5010 5030	Device identifier for "read-in" in 1st/3rd V.24 (RS232C)							
5011 5031	Number of stop bits		Transfer format for "read-in" from 1st/3rd V.24 (RS232C): Odd parity	With parity *)	Baud rate			
5012 5032	Device identifier for "read-out" in 1st/3rd V.24 (RS232C)							
5013 5033	Number of stop bits		Transfer format for "read-out" from 1st/3rd V.24 (RS232C): Odd parity	With parity *)	Baud rate			
5014 5034	X_{ON} start character 1st/ 3rd V.24 (RS232C) (value e.g. 11_H)							
5015 5035	X_{OFF} start character 1st/3rd V.24 (RS232C) (value e.g. 13_H)							
5016 5036	Special bits 1st/3rd V 24 (RS232C): Start without X_{ON}	Program start with LF	Block end with CR LF	Output in EIA code	Stop with end of transfer character	Analyse readiness	No leader and trailer	Read-in program from System 3/8
5017 5037	Special bits 1st/3rd V 24 (RS232C)					Output of MDS not equal to zero	Cancel program with REORG	Time watchdog OFF
5018 5033	Device identifier for "read-in" in 2nd/4th V.24 (RS232C)							
5019 5039	Number of stop bits		Transfer format for "read-in" from 2nd/4th V.24 (RS232C): Odd parity	With parity *)	Baud rate			

*) *When reading out, the NC adds and when reading in it checks the 8th data bit for even parity in the case of ISO code and for odd parity with the EIA code. Additional parity generation and checking can be selected with the "with parity" setting data. This additional parity is not suitable for ASCII data transmission, however, owing to the nested parity. The additional parity can therefore be used only with ISO and EIA-compatible devices. See SINUMERIK System 800, Universal Interface, Planning Guide, Section 4.3.*

SD No.	Bit No. 7	6	5	4	3	2	1	0
5020 5040	Device identifier for "read-in" in 2nd/4th V.24 (RS232C)							
5021 5041	Number of stop bits		Odd parity	With parity *)	Transfer format for "read-out" from 2nd/4th V.24 (RS232C): Baud rate			
5022 5042	X_{ON} start character 2nd/4th V 24 (RS232C) (value e.g. 11_H)							
5023 5043	X_{OFF} start character 2nd/4th V 24 (RS232C) (value e.g. 13_H)							
5024 5044	Special bits 2nd/4th V 24 (RS232C): Start without X_{ON}	Program start with LF	Block end with CR LF	Output in EIA code	Stop with end of transfer character	Analyse readiness	No leader and trailer	Read-in progr. from System 3/8
5025 5045	Special bits 2nd/4th V 24 (RS232C)					Output of MDs not equal to zero	Cancel program without REORG	Time watchdog OFF
5026 5046	EIA code for "@" (value e.g. $6D_H$)							
5027 5047	EIA code for ":" (value e.g. $46D_H$)							
5028 5048	End of transfer character (value e.g. 03_H)							
5029 5049	EIA code for ":" (value e.g. $7D_H$)							

*) *When reading out, the NC adds and when reading in it checks the 8th data bit for even parity in the case of ISO code and for odd parity with the EIA code. Additional parity generation and checking can be selected with the "with parity" setting data. This additional parity is not suitable for ASCII data transmission, however, owing to the nested parity. The additional parity can therefore be used only with ISO and EIA-compatible devices. See SINUMERIK System 800, Universal Interface, Planning Guide, Section 4.3.*

SD No.	Bit No. 7	6	5	4	3	2	1	0
5050	EIA Code for "[" 1./2. V24 (RS 232)							
5051	EIA Code for "]" 1./2. V24 (RS 232)							
5052	EIA Code for "," 1./2. V24 (RS 232)							
5060	EIA Code for "[" 3./4. V24 (RS 232)							
5061	EIA Code for "]" 3./4. V24 (RS 232)							
5062	EIA Code for "," 3./4. V24 (RS 232)							

Setting data 1./3.V24 Setting data 2./4. V24	5010/30 5018/38	5011/31 5019/39	5012/32 5020/40	5013/33 5021/41	5014/34 5022/24	5015/35 5023/43	5016/36 5024/44
Device	Relevant bit pattern						
PG685 w. CP/M86 1200 Baud	0000 0000	1100 0100	0000 0000	1100 0100	xxxx xxxx	xxxx xxxx	xx1x 1xxx
GNT reader (Opt. B02/B03)	0000 0000	1100 0111	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx	0000 0000
Fanuc hand-held reader	0000 0001	1100 0110	xxxx xxxx	xxxx xxxx	0001 0001	1001 0011	0000 0000
PT80 300 Baud	0000 0000	1100 0010	0000 0000	1100 0010	xxxx xxxx	xxxx xxxx	0000 0000
PT88 9600 Baud V.24	xxxx xxxx	xxxx xxxx	0000 0000	1100 0111	xxxx xxxx	xxxx xxxx	0000 0000
WS 800	0000 0000	1100 0111	0000 0000	1100 0111	xxxx xxxx	xxxx xxxx	xx1x 1xxx
PG 675/685 with Step 5 and 9600 Baud (only 3rd interface)	0000 0100	xxxx x111	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx
VT 340 (only 1st/3rd V.24)	0001 0010	xxxx xxxx	0001 0010	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx
VT printer (only 1st/3rd V.24)	0001 0011	xxxx xxxx	0001 0011	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx
Teleservice (only 1st/3rd V.24	0001 0001	xxxx xxxx	0001 0001	xxxx xxxx	xxxx xxxx	xxxx xxxx	xxxx xxxx

x .... irrelevant

Baud rate setting	
Bit pattern Bits 3 to 0	Baud rate
0000	110 baud
0001	150 baud
0010	300 baud
0011	600 baud
0100	1200 baud
0101	2400 baud
0110	4800 baud
0111	9600 baud
1000	19200 baud

Device identifiers	
Bit pattern	Device type
0000 0000	Line controlled devices (RTS line)
0000 0001	X_{ON}/X_{OFF} controlled devices
0000 0010	
0000 0011	SIEMENS NC programming workstation PD...PG
0000 0100	PG 685/730/750 with STEP 5
0001 0010	Terminal emulation
0001 0011	VTE-LPT
0001 0001	Teleservice

In the case of the SINUMERIK 880 GA2, four interfaces are available, allowing data transfer with external devices.

Interfaces:

Interface 1	OP CPU	X121 (standard)
Interface 2	OP CPU	X131 (option)
Interface 3	COM CPU	X121 (standard)
Interface 4	COM CPU	X131 (option)

The four interfaces are combined in two groups. In each case two interfaces from different groups can be operated simultaneously. The interfaces in any one group cannot be operated at the same time.

Groups:

Group 1 Interfaces 1 + 2
Group 2 Interfaces 3 + 4

The simultaneous inputting of different data types does not have any interference effect. The following applies when the same data types are input simultaneously:

In the case of part programs, all of the first program to have been started is stored in the memory. The other interfaces wait with their programs and are continued in succession. In the process, an interface may possibly be stopped for longer than the internal timer (1 min). Consequently, the timer can be switched off by means of the setting data.

If the other data types (e.g. machine data, TO, ZO, etc.) are input simultaneously, the last value to have been entered is held in the memory.

9.6.16 Channel-specific NC setting bits

540*	SD No. Bit No.						540*
7	6	5	4	3	2	1	0
						Spindle converter	Axis converter activated

Bit 1: Can only be activated via PLC (function same as axis converter).

Bit 0: The axis converter is used to convert the names of axes during processing without having to make other changes in the part program. Thus the axes X and Z may be programmed in the part program but axes X1 and Z1 processed.
The axis converter does **not** work for:
- Radius and chamfer
- Angle
- Spindle

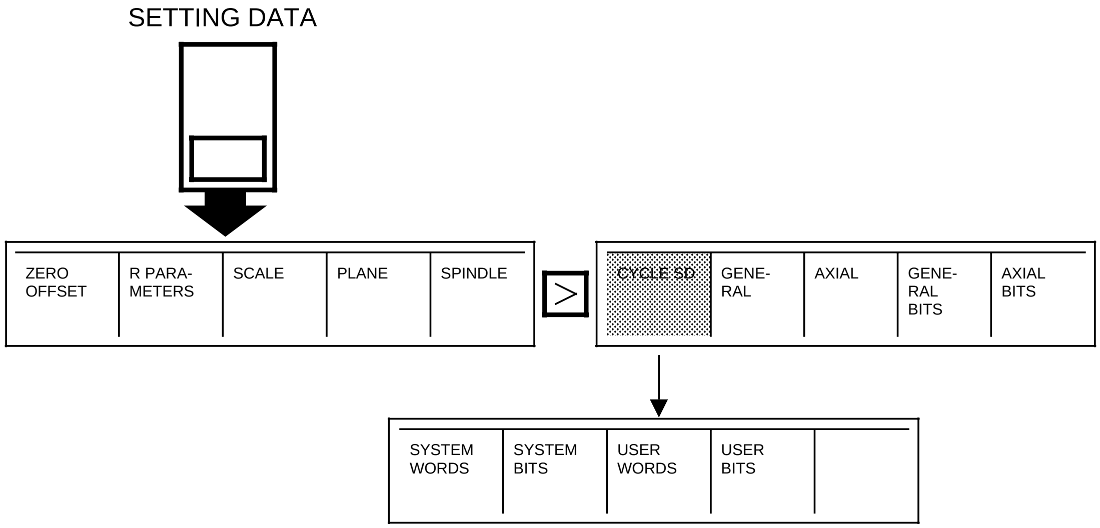
9.6.17 Axial NC setting data bits

560*	SD No. Bit No.						560*
7	6	5	4	3	2	1	0
				Scale factor active in simulation	Scale factor active in machining	Rapid traverse override not active	Feedrate override not active

Bit 3 The scale factor (G51) is active in the simulation
Bit 2 The scale factor (G51) is active for machining (channels 1 to 8 (out of 16))
Bit 1 Override switch no longer effective on rapid traverse in the relevant axis
Bit 0 Override switch no longer effective on feedrate in the relevant axis

9.7 Cycle setting (cycle SD)

Cycle SDs are used, as are cycle MDs (see Section 9.5), to store values in a safe memory.



Cycle SDs are available separately for every channel (channel-specific). The cycle SDs are divided into two areas as follows (system area Siemens and user area).

Channel-specific cycle SD:

0 ⋮ 99	Words system (Siemens)
400 ⋮ 499	Words user
800 ⋮ 849	Bits system (Siemens)
900 ⋮ 949	Bits user

The cycle SDs are deleted in installation mode with the softkey "FORMAT USER M". The input or modification of cycle SDs on the NC operator panel can be disabled with the key switch (NC MD 5051 bit 0).

The cycle SDs reserved for SIEMENS are assigned exclusively by SIEMENS Nuremberg.

The cycle SDs can be read and written with the following CL 800 comands (see also Programming Guide):

@ 313	Read cycle SDs	Word
@ 314	——— " ———	Byte
@ 315	——— " ———	Bit
@ 413	Write cycle SDs	Word
@ 414	——— " ———	Byte
@ 415	——— " ———	Bit

The cycle SDs can be read by the PLC with FB 61 and written by the PLC with FB 62.

10 Machine Operation/Service Data

10.1 Jog mode

Preconditions:

- All axis setpoint cables plugged in
- Correct control direction
- Position control loops closed
- Correct gains

The alarms below may prevent the axes from being traversed:

Alarm No.	Explanation
2000	"EMERGENCY STOP"
148* **152*** **188*** **192***	Hardware or software limit switch approached Hardware: • Test using interface test • Software limit switches reached (only after reference point approach). Limits via NC MD machine data 224*, 228*,232*,236*
168*	Servo enable for a moving axis cancelled by interface unit
156*	Set speed too high. Activated via NC MD machine data 264*
112*	Clamping error Axis not in position. Activated via NC MD machine data 372*
116*	Contour monitoring (initiation via NC MD 332*, 336*)
132* **136*** **140*** **144***	Hardware measuring-circuit monitoring Measuring-circuit signal monitor has responded for axes or spindle (can only be cleared via hardware reset, POWER ON)

In addition, the following signals are required for jog mode:
(no alarms activated)

- Feed enable X, Y, Z, 4 - 24
- Feed enable, overall
- No axis inhibit X, Y, Z, 4 - 24
- Servo enable X, Y, Z, 4 - 24
- No follow-up mode X, Y, Z, 4 - 24

Interface test

** = Axes 1-24*

Should there be no feed and servo enable signals, the "Feed Hold" LED lights up after the direction key has been actuated. The following signals must not be present if traversing is to be performed at the input speed influenced by the feedrate or rapid traverse override switch:

- F external (feedrate from PLC)
- Feedrate reduction ratio 1:100
- Testing of all jog functions:
  - Limit switches
  - External deceleration (rapid traverse reduction)
  - Feedrate override
  - Incremental dimension
  - Reference point approach

10.2 Progammed operation

In this case only the principal function is to be tested so that programs can be used as an aid to optimization.

The following data and signals can still prevent programmed operation:

- Read-in disable (DL 7 to DR 10)
- NC START=1 / NC STOP=0 (DR 2)
- Disable of NC START (DW 11)
- NC MD 106*
- Reference point not approached or NC MD 5004 bit 3
- NC MD 548*, 550*, 552*
- NC START inactive (DL 16)

Check whether the axes can traverse via the program memory.

10.3 Functional test with NC test tape

For testing the following functions:

- Block display
- Zero offset
- Interpolation
- Transfer (overstore)
- Tool offsets
- All auxiliary functions (M, S, T, D, H)
- Thread cutting
- Single block, deletable blocks, program stop
- Program end
- Subroutines and R parameters
- Program editing
- Program memory operation
- Blanking of programs
- Simultaneous reading-in of programs during processing.

The program and tape are to be provided by the machine manufacturer.

10.4 Axis and spindle service data

For drive optimization and fault diagnosis, it must be possible to view the data transmitted from the NC to the axes or spindles and from the axes or spindles to the NC.

Caution!

Service values are displayed at twice the value, i.e. in units of position control resolution (e.g. a following error display of 2000 with position control resolution of 0.5 μm is equivalent to an actual following error of 1 mm).

10.4.1 Axis service data

The following service data for axes are displayed:

- Following error in units of position control resolution
  Difference between setpoint and absolute actual position.
- Absolute actual position in units of position control resolution
  Actual position of axes at machine.
- Setpoint (command value) in units of position control resolution
  Specified value determined by the controller on the basis of the program or set position entered manually. The setpoint and absolute actual position are usually identical (at rest the difference (following error) can be offset by the drift compensation).
- Set speed in VELO
  Digital value determined by the controller (for maximum value see NC MD 268*). It is converted at the measuring-circuit module to an analog value (0 V to 10 V) and output as a setpoint to the drive.
- Partical actual value in units of position control resolution
  Pulses from measuring system per scanning cycle (standard 2.5/5/10 ms).
- Partical actual value in units of position control resolution
  Set part position values output by the interpolator to the position control per IPO cycle (standard 20 ms).
- Contour monitoring in units of position control resolution
  This value is used to indicate the current contour deviation (fluctuations in the following error due to compensating operations at the speed controller as a result of load changes).
- Absolute compensation value in units of position control resolution
  The resulting value from interpolatory and temperature compensation is displayed here.
- Service number
  With certain malfunctions a service number appears here by way of exact diagnosis.

List of service numbers

- **Absolute encoder errors**

The following list contains all possible error numbers and potential causes of the error in each case, assuming that the connection between the encoder and absolute module is in perfect condition:

Error number:	**1...50, 93...195, 240...253**
Meaning: Remedy:	Absolute encoder and/or absolute module defective • Check absolute encoder and absolute module for damage • Check that connectors are properly seated
Error number:	**51...70**
Meaning: Remedy:	Absolute module and/or connection between absolute module and measuring circuit defective • Check absolute module for damage • Check that connector between measuring circuit and absolute module is properly seated
Error number:	**71...80, 200...230**
Meaning: Remedy:	Absolute module defective / battery flat ("battery alarm" output after 10 min) • Check absolute module for damage • Check that encoder connector is properly seated
Error number:	**81...86**
Meaning: Remedy:	Connection between absolute module and measuring circuit defective • Check absolute module for damage • Check that connector between measuring circuit and absolute module is properly seated

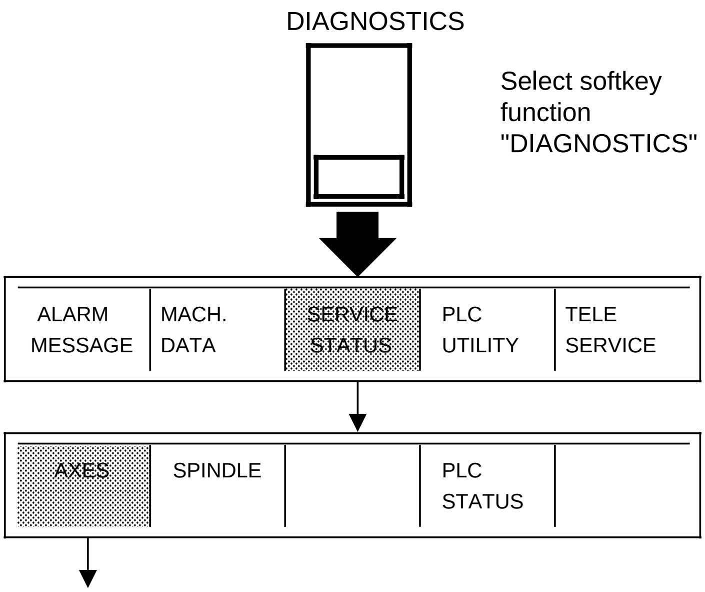
- **Parameterization errors axis/spindle**

Error number:	**300**
Meaning: Remedy:	Incorrect sampling ratio Check MDs 155, 160 - 163, 168, 1396*, 466*
Error number:	**308**
Meaning: Remedy:	Incorrect increment weighting Check MDs 364*, 368*, 1208*, 1212*, 455*, 456*, 524*
Error number:	**309**
Meaning: Remedy:	Incorrect actual value resolution HMS Check MDs 1116*, 1204*, 458*

Error number:	**314**
Meaning: Remedy:	Illegal servo gain (K_V) factor Check NC MDs 252*, 260*, 435* to 442*
Error number:	**316**
Meaning: Remedy:	Illegal module value / axis is not a rotary axis Check NC MD 344*, 564* bit 5
Error number:	**317**
Meaning: Remedy:	Sampling interval FS sampling interval LS Check NC MD 1396*, 466*
Error number:	**318**
Meaning: Remedy:	Incorrect leading/following axis configuration (leading axis/spindle missing or not on same servo CPU) Check NC MD 200*, 400*, 461*
Error number:	**319**
Meaning: Remedy:	Illegal maximum speed (scale overflow) Check NC MD 264*, 268*, 260*, 403* bis 410*, 419 bis 426*, 468*
Error number:	**321**
Meaning: Remedy:	Feedforward control parameterization defective Check NC MD 312*, 1124*, 1260*, 465*
Error number:	**322**
Meaning: Remedy:	Incorrect system clock setting Check NC MD 168
Error number:	**323**
Meaning: Remedy:	Invalid position control resolution or position control resolution (spindle) position control resolution (axis) Check NC MD 1800*, 524*
Error number:	**324**
Meaning: Remedy:	Incorrect C axis assignment: axis does not exist on same servo CPU, no spindle encoder available or mode group (spindle) mode group (axis) Check NC MD 200*, 400*, 461*, 524*

Error number:	**325**
Meaning: Remedy:	Position control sampling interval C axis spindle Check NC MD 1396*, 466*
Error number:	**326**
Meaning: Remedy:	Incorrect measuring gearbox Check NC MD 364*, 368*, 1208*, 1212*, 455*, 456*, 524*
Error number:	**327**
Meaning: Remedy:	Both measuring systems are defined as systems with distance-coded reference marks Check NC MD 1808* bits 4 and 5, declare only one measuring system
Error number:	**504**
Meaning: Remedy:	Servo gain (K_V) factor too small Check NC MD 435* to 442*
Error number:	**515**
Meaning: Remedy:	Invalid position control resolution Check NC MD 1800*, 524*
Error number:	**520**
Meaning: Remedy:	Incorrect C axis assignment: set axis No. invalid Check NC MD 461*
Error number:	**526**
Meaning: Remedy:	Multgain too small Check NC MD 468*
Error number:	**529**
Meaning: Remedy:	Illegal acceleration time constant Check NC MD 419* to 426*, 478* to 485

Selection of axis service data



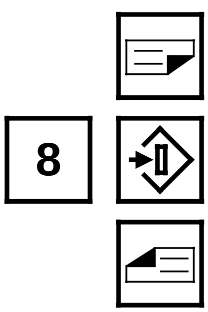
Select softkey function "DIAGNOSTICS"

Select "SERVICE STATUS" in new softkey menu

Select "AXES" in new softkey menu

When the softkey is pressed, the above-mentioned axis-specific values are displayed:

Display:	Axis 1	1st axis
Display:	Axis 2	2nd axis
.		.
.		.
.		.
Display:	Axis 24	24th axis



The "Page forwards" key makes it possible to page to the next axes.

Entering "8" and actuating the search key makes it possible, for example, to select axis 8 directly.

The "Page backwards" key makes it possible to page back to the next axes if necessary.

10.4.2 Spindle service data

For optimization and fault diagnostics purposes, it is possible to view the current spindle values.

- Set speed (VELO):
  Digital voltage output by NC at measuring circuit.
- Set speed (rev/min):
  Value input by the user;
  e.g.: Input S 1000   Display: set speed 1000 (rev/min)
- Actual speed (rev/min):
  The pulses from the spindle encoder are evaluated by the NC and indicated as the speed in rev/min.

- Set position:
  The spindle position specified by the user in degrees is converted by the NC to the appropriate number of units [MS];
  e. g.: 0° = 0 with position control resolution 0.5 10^{-3}
  180° = 180 000
  359° = 359 000

- Actual position:
  The pulses from the spindle encoder are evaluated by the NC and displayed;
  e.g.: 90 000 units [MS] spindle position= 90° .

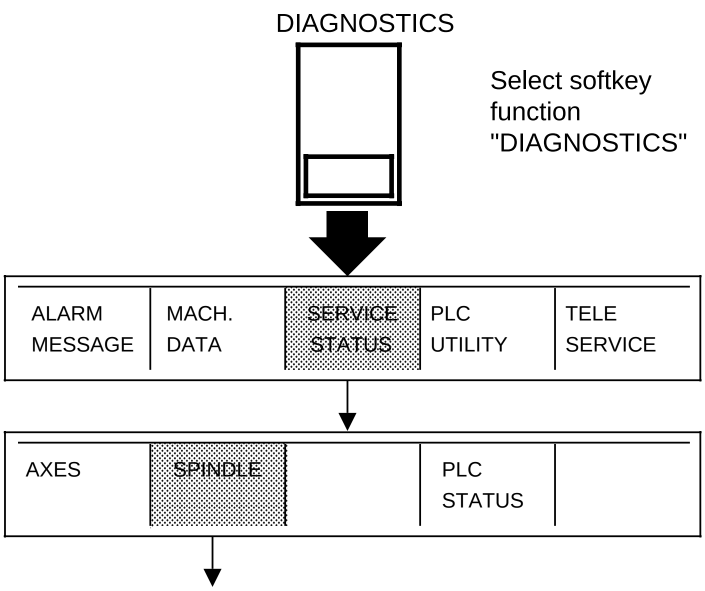
- Following error:
  Difference between the set position and the actual position in units of position control resolution. Only useful with controlled operation.

- Override:
  The position of the spindle override switch is indicated.

- Gear stage:
  The current gear stage is indicated. (DB 31 DR K+1 bits 0 to 2).

- Service number
  With certain malfunctions a service number appears here by way of exact diagnosis. The service numbers are listed with the text describing the axis diagnostics.

Selection of spindle service data:

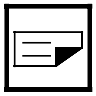


Select softkey function "DIAGNOSTICS"

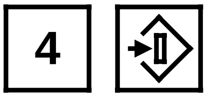
Select "SERVICE STATUS" in new softkey menu

Select "SPINDLE" in new softkey menu

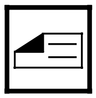
”Page forwards/backwards” or direct selection of a specific spindle
(up to 6 spindles in the case of the SINUMERIK 880 GA2) as described in Section 10.4.2.



The "Page forwards" key makes it possible to page to the next spindle.



Entering "4" and actuating the search key makes it possible, for example, to select spindle 4 directly.



The "Page backwards" key makes it possible to page back to the next spindle if necessary.

11 Function Description

11.1 Execution from external

The EXECUTION FROM EXTERNAL

function is an Option.

11.1.1 Corresponding data

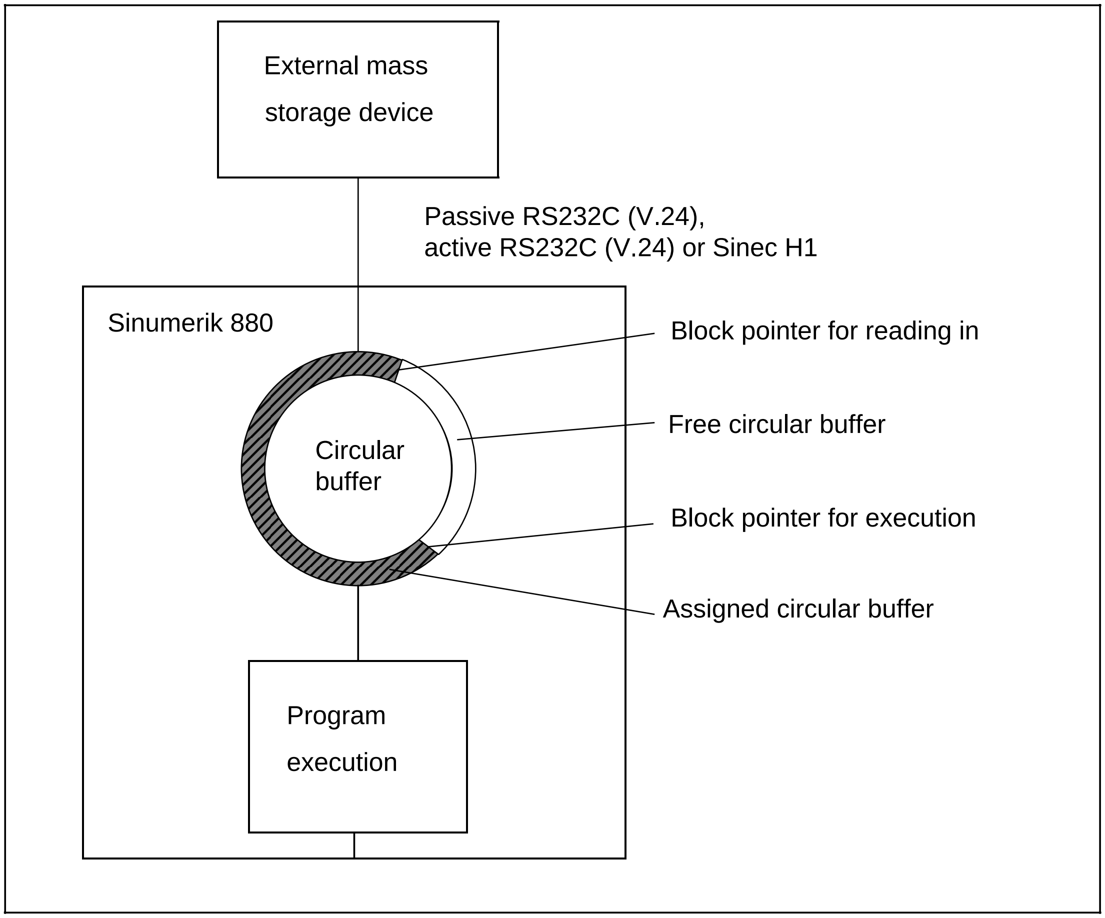
- NC MD 30 — No. of sectors of executing memory
- NC MD 130* — Serial interface for execution from external (channel-specific)
- NC MD 5148 - 5151 — Logic partner receiver
- NC MD 5152 — Location receiver

- DB 10-25 DW2 bit 15 — Execution from external (PLC NC, activation bit)
- DB 10-25 DW14 bit 7 — Execution from external (NC PLC, softkey selection)

11.1.2 Function description

The "execution from external" function allows long part programs to be read into the control from an external mass storage medium via a serial interface (RS232C (V.24)) or computer link and executed at the same time.

In this context the part program is input into a circular buffer block by block, from where it is executed directly.



The transferred program blocks are loaded in the circular buffer and executed from there. During program execution, the memory area that has been processed is cleared again and automatically loaded with new blocks.

If the selected program already exists in the memory, alarm 2045 is issued.

The memory area of the circular buffer is located in the normal part program memory. Enter the number of sectors the circular buffer is allowed to use in NC MD 30. The user must make sure that the sectors are free if he wishes to utilize this function.

The interface specified with NC MD 130* is initiated with NC START; in the case of file transfer/computer link this leads to execution of the "Transfer of NC data to NC on user initiative" file transfer dialog.

11.1.3 Function sequence

11.1.3.1 Selection

- By pressing the "execution from external" softkey in the "program modification" menu (DB 10-25, DR 14, bit 7). Press the softkey again to deselect the function. This applies to the RS232C (V.24) **and** computer link interfaces on machine operator initiative.
- By setting the channel-specific VDI signal "execution from external" (DB 10-25, DL 2, bit 15) in the case of automatic control by a host computer in conjunction with the PLC user program.

The selection is confirmed by the "EXT" message in the status line (1st line of screen). Selection by softkey is retained after POWER ON.

11.1.3.2 Start

- As before, the program to be executed is selected in the automatic basic display and initiated with NC Start. A main program or subroutine can be selected. This applies to the RS232C (V.24) **and** computer link interfaces on machine operator initiative.
- In the case of control by a host computer, the program is initiated by the "NC program selection with start enable" message (user message**/**function to be implemented by user).

If the interface cannot be initiated or the selected program is already being executed in another channel, alarm 2044 "error execution external" is displayed and processing is aborted. In the case of selection by host computer, the user must evaluate this alarm and report it to the host computer where applicable.

11.1.3.3 Reading in

Input by RS232C interfaces

See Programming Guide for program format.

If the selected program is read in by an RS232C (V.24) interface, execution begins when the first program block is in the circular buffer.

Users are recommended to program a dwell time (approx. 3 sec) first so that a buffer is created between the read-in and execution pointers. If a different program or other data are read in, these are input as usual. Processing is halted in this event.

If an error occurs during reading in, the read-in routine is aborted. Execution continues, however, as long as the circular buffer still contains program blocks.

Input by file transfer

The "read-in with user requests" frame sequence is executed by file transfer.

The request frame (TDMPF, TDSPF) is sent o the "logic partner receiver" specified in machine data 5148-5151 (ASCII values, standard "FLR__"). The module ("location receiver") to which the logic partner is connected is specified in machine data 5152 (where 11H = 1st CP, 12H = 2nd CP; 11H is standard).

Here again, programs other than those requested can be read in first by the host computer (e.g. subroutines belonging to the main program). However, these must be transferred by the host computer in the same file transfer sequence as the program that is to be executed.

11.1.3.4 Execution

General

Execution commences as the selected program is being read in. Processed program blocks are deleted while execution is still continuing. When reading in, the program is restricted to the number of sectors (507 bytes each) specified in machine data 30. The standard setting here is 10 (= approx. 5 Kbytes). A minimum length of 5 sectors (approx 2.5 Kbytes) is considered necessary to ensure troublefree program execution. Since the program is read into the normal part program memory, the required number of sectors must be kept free, otherwise only the available memory area will be used. If insufficient memory capacity is available, execution is halted.

Program jumps

Forward jumps in the program are allowed. If the jump destination has not been read in yet, execution is halted until it is. Similarly, execution is halted if the specified memory area is filled before the jumper destination is read in. Return jumps and static jumps abort execution. In this event, alarm 3012 "block not in memory" is displayed.

Subroutine calls

Subroutine calls are allowed as before. The called program must already be in the part program memory or user memory submodule.

Dilemma: execution faster than reading in

If a part program is executed very quickly (blocks processed close to block change time), the circular buffer might become empty because the serial interface or computer link is unable to read in the next blocks fast enough. In this event, the speed is reduced at the end of the block, until the axes are at zero speed where applicable.

The following measures can be taken in order to avoid this speed drop:

a) Interrupt execution at a suitable place with read-in disable, NC Stop or feed hold and read out the number of characters that have not yet been processed and are still in the circular buffer via the PLC (FB 61). Where appropriate, execution can be halted until the number of characters required for the next program section has been read into the circular buffer.

b) The feedrate can be reduced.

11.1.3.5 Interrupting/aborting execution

Interrupting

Execution is interrupted with NC Stop or read-in disable; program blocks continue to be read in until the circular memory is full.

Execution can be resumed with NC Start or read-in enable.

Aborting

Execution is terminated with Reset or M02/M30. This stops the interface and the rest of the program is deleted from the memory.

11.1.3.6 Remarks

- Block search with calculation (Start-1) is still allowed. As with NC Start, the interface is started and commences with the block search.
- External simulation of a program is not allowed.
- The current block display shows only the current and next blocks.
- As usual, comments are displayed on the screen in the alarm line.
- Once the program has started, no editing is allowed at the control.
- If a decoding error occurs during execution, the defective block can be viewed under compensation block. No editing is allowed here either.

11.2 Spindle control

The SPINDLE CONTROL function is an Option with the M software version.

11.2.1 Corresponding data

- MD 131 - 146 (Spindle override)
- MD 4000 - 4890 (Spindle data)
- "Analog spindle speed" Option
- MD 5200 bits 0 - 7
- MD 5210 bits 1 and 7
- MD 5240 Spindle position control resolution
- "Spindle package" Option

11.2.2 S analog (M3, M4, M5)

Spindle commissioning is explained in Section 6.1.6 and is not therefore set out once more at this point.

In the case of the SINUMERIK 880, 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 11.2.4). Spindle data for up to 8 gear stages and additional monitoring functions are stored in the controller.

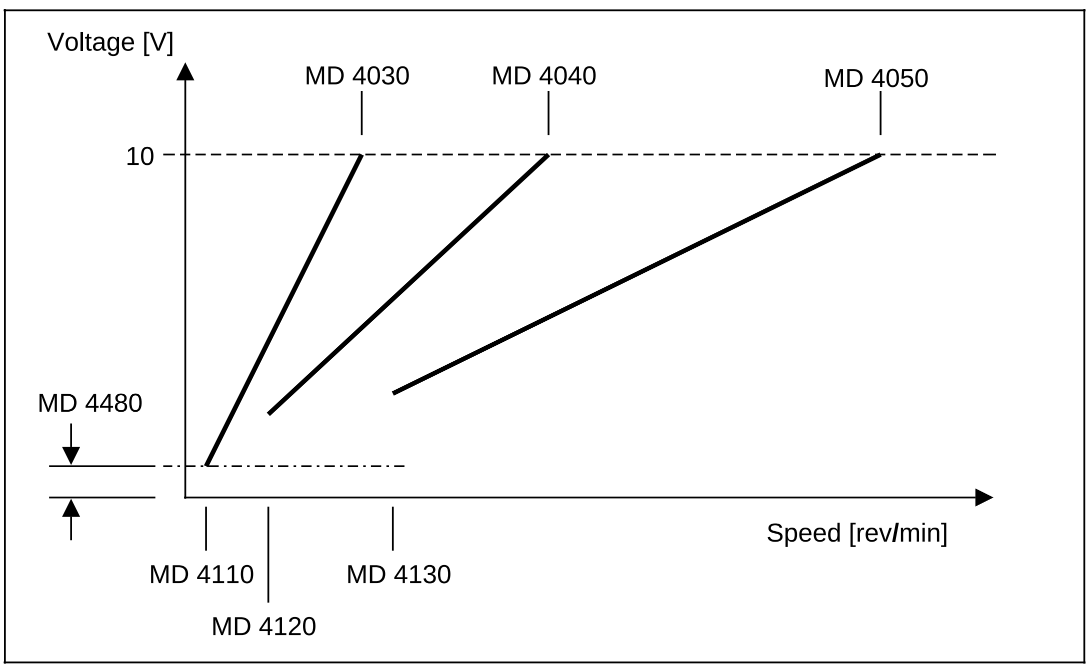
The function "Automatic gear selection" is also implemented in the NC. The "Change gear" interface signal is output by the NC as a function of the programmed S value (provided that the spindle disable signal is not and the servo enable signal is present) and the "specified gear" defined on the basis of the machine data is applied to the interface. "Spindle disable" is then output by the PLC. The spindle motor decelerates to zero speed. The "PLC spindle control" and "Reciprocation speed" signals must be output by the PLC and the "Spindle disable" signal reset. The direction of rotation is switched by the PLC to facilitate engaging of the gear. Once the gear has been engaged, the PLC must set the SPINDLE DISABLE interface signal and then report the interface signals "Change gear", "PLC spindle control", "Reset oscillation speed", "Set spindle disable" and the new "actual gear" to the spindle. After the PLC has enabled the spindle (reset spindle disable), acceleration to the new speed is performed.

Interface signals: DB31 spindle-specific signals

Note:

Automatic gear selection is only possible with the S analog function.

The gear stages (8 max.) are precisely defined by means of the minimum and maximum speeds of each gear stage.



The gear stage is output on the basis of the lowest switching frequency, i.e. if the speeds of the individual gear stages overlap, a new gear stage is only output when the programmed S value is no longer possible in the selected gear stage.

In view of the fact that not all spindle drive units include a ramp-function generator, such a generator was integrated into the 880 (units 1 ms) in each case. The following enable signals are required for spindle setpoint output:

DB 31 Spindle-specific signals

- DL_{K+1} Bit 6 = 1 "Servo enable"
- DL_{K+1} Bit 5 = 0 "Specify setpoint 0"
- DW_{K+3} all bits = 0 "Spindle disable"
- Setting data "Spindle speed limitation"
- DB31 DL_{K+2} "Spindle channel No."
- Switchover from command channel "S external" function must not be set.

11.2.3 M19 (oriented spindle stop)

The oriented spindle stop function (M19 S...LF) is intended to prevent additional external hardware requirements if the spindle is to be stopped in a specific position for a tool change or to engage gear.

A basic distinction is made with M19 between:

- NC-controlled spindle positioning
- PLC-controlled spindle positioning

The simultaneous use of both types of positioning is not possible and would give rise to control malfunctions. In this case the ROD encoder is used not only for speed control purposes (G95) 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 (M3, M4, M5) the spindle is controlled by the NC; only with M19 is the position control loop closed by the NC. The pulses from the ROD encoder act as actual position values.

The oriented spindle stop function is activated in the part program with "M19". The target position is stored as a setting data which can be set with manual input or "M19 S..." programming in degrees.

The positioning range is 0.01 to 359.99 degrees. Positioning is carried out in the specified direction of rotation (M3, M4) or from rest over the shortest distance.

The spindle may also be positioned using external devices if the "oriented spindle stop" Option has not been set. In this case, M19 is output to the PLC as a normal auxiliary function (also a static or dynamic flag). MD 5200 bit 2 "pulse generator available" has no significance.

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.

Sequence A:

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.

Sequence B:

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 (A and B):

- M19 S... must be programmed in a special block without axis movements
- Orientation is performed in the specified direction of rotation (M03/M04)
- M19 is possible from rest (shortest travel)
- Oriented spindle stop is initiated at the start of the block
- M19 is interrupted or terminated by means of
  - EMERGENCY STOP (Alarm 2000)
  - Reset (depending on machine data)
  - Program end (M02/M30) (depending on machine data)
  - Spindle inhibit (DB 31 DW $_{K+3}$)
  - Servo enable (DB 31 DW$_{K+1}$)
  - Measuring-circuit error in spindle or an axis
  - Errors resulting in shutdown of all axes
  - Cancelling of NC Ready 2
  - Acknowledge M19 (DB 31 DW $_{K+2}$)
  - Selection of M3, M4
- M19 can be selected in "MDI-AUTO", "JOG" or "AUTOMATIC" mode, as well as overstore
- M19 is output as an auxiliary function at the PLC interface

Special features of sequence A (NC MD 520* bit 5 = 0)

- A block change is only performed on completion of the M19 function (acknowledge M19)
- Simultaneous traversing of the axes is not possible.

Special features of sequence B (NC MD 520* bit 5 = 1)

- M19 is modal, even over several blocks. If M19 is selected but the spindle servo enable signal is missing, the next block is executed nonetheless. M19 is not executed until the servo enable is issued.
- The block change is performed after a delay of one PLC cycle.
- In the subsequent blocks, axes can be moved or a tool change can be performed at the same time as positioning or closed-loop position control.
- With positioning active (M19), the direction of rotation (M03/M04) must not be reversed, otherwise positioning is performed from an undetermined direction.
- If the PLC detects auxiliary function M19, it can prevent a block change by cancelling the READ-IN ENABLE.
- If M19 is selected again before a preceding M19 has been concluded, the control function refers to the new spindle position and the NC approaches this position over the shortest distance irrespective of the specified direction of rotation; the spindle travel is less than 180°, irrespective of the control characteristic.

Time sequence:

1. Deceleration of the spindle from the programmed speed to the cutoff speed (NC MD 427*) with reference to the ramp characteristic specified by the acceleration time constant (NC MD 419* - 426*).
2. When the cutoff speed is reached, the M19 position is approached with interpolation according to the characteristic (NC MD 478*, acceleration time constant with position control).
3. Output of "spindle position reached" message (DB31 DL_K bit 4) to the PLC if the actual spindle position has fallen short of the tolerance threshold in MD 4430.
4. M19 is considered to have been terminated when the PLC issues the "acknowledge M19" signal (DB31 DR_K+2 bit 2) or in the event of M3, M4. In this case the position control is disengaged but the spindle servo enable relay does not drop out (spindle can drift).

PLC selection

With PLC-controlled positioning, the positioning operation is triggered by the PLC by transmitting the "PLC spindle control" and "spindle positioning" signals to the spindle. The setpoint is specified in NC MD 452* "spindle position with ext. M19".

The positioning operation is terminated when the PLC transfers the "acknowledge M19" signal to the spindle with the "position reached" signal or when M3/M4 is executed. Positioning from zero speed is allowed.

In other respects the sequence is identical to NC-internal spindle positioning.

Note:

The two variants (M19 via NC and M19 via PLC) cannot be used at the same time.

M19 from rest

The spindle will always approach the programmed position from rest over the shortest path. The "Set direction of rotation clockwise" signal (DB 31 DR_K+2 bit 7) has no significance.

Positioning accuracy

The target position is programmed in 0.01 degrees with decimal point. The accuracy of this position depends on the gain, gear stage and drift. The maximum achievable accuracy depends on the pulse generator being used.

Theoretically, the positioning accuracy could be improved by increasing the gain. However, it must be borne in mind that this would increase the spindle's tendency to oscillate.

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 only aborted by means of the "Acknowledge M19" signal from the PLC or alarms cancelling NC Ready 2 or EMERGENCY STOP.

Acknowledge M19

If the "Spindle position reached" signal is output during positioning by the NC, the PLC must cancel the "Acknowledge M19" signal or M3/M4 must be executed if function M19 is to be terminated.

The signal "Acknowledge M19" not only aborts spindle positioning but the spindle is also stopped (like M05) and a **new** M03 or M04 must be programmed in the part program so that the spindle can run up to the previous programmed S value.

The "Acknowledge M19" signal is also active without "PLC spindle control".
M functions M3 and M4 also terminate positioning, irrespective of whether or not the position has already been reached.

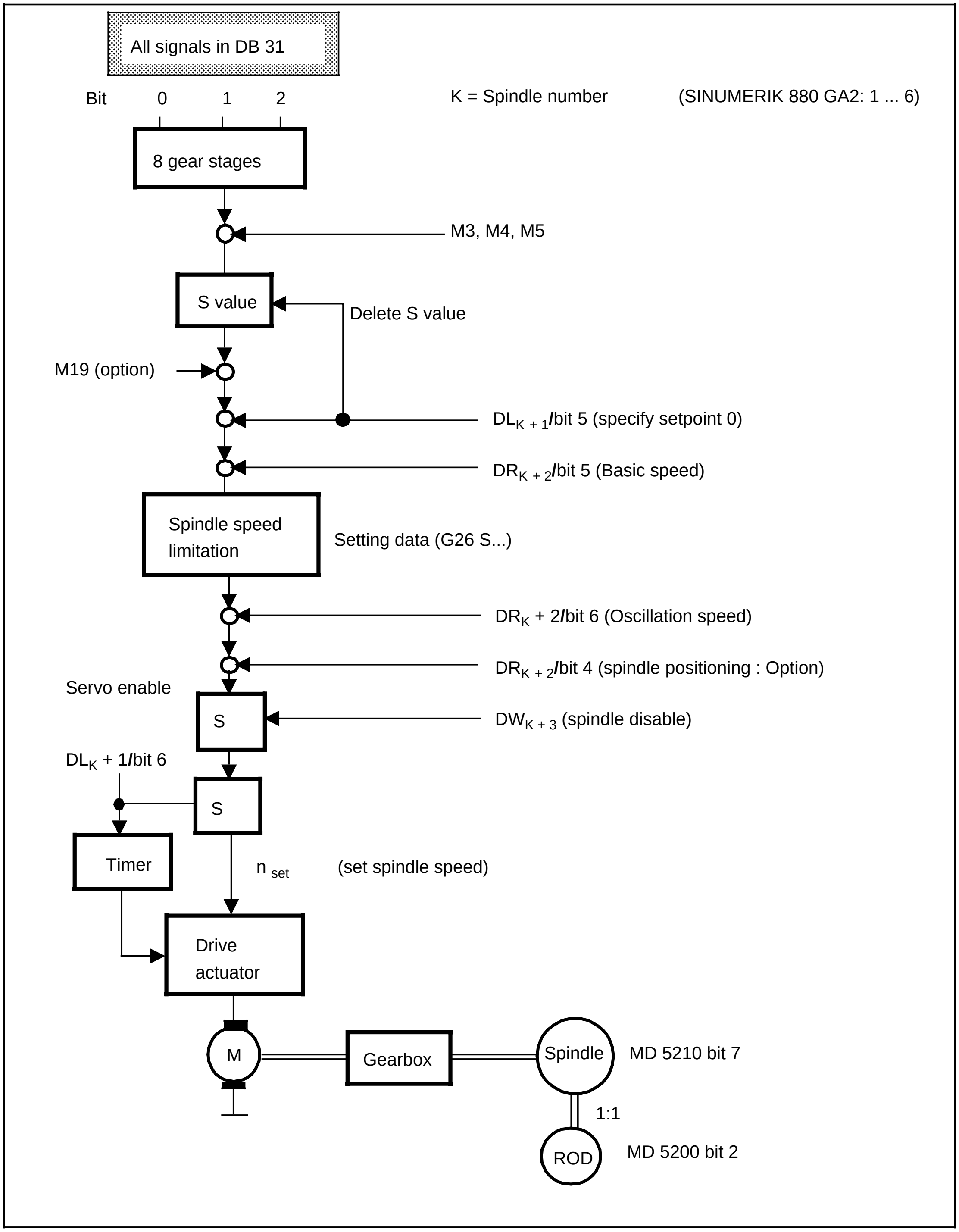
Position control direction with M19

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° ahead of the programmed position, the spindle exhibits severe oscillation about this position.
In this case NC MD 521* bit 1 must be inverted.

11.2.4 Spindle influencing by PLC

The function diagram overleaf 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.

The signals "Set direction of rotation clockwise", "Oscillation", "Basic speed", "Spindle positioning" and "Spindle resynchronization" are only active in conjunction with the "PLC spindle control" signal.



Ways of intervening in the PLC spindle control by PLC interface signals

11.3 Approach to reference point

11.3.1 Corresponding data

• 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)
• Signal "Reference point reached"	(DB 32 DL_K bit 4)
• Signal "Deceleration reference point approach"	(DB 32 DL_{K+1} bit 4)

Directly related:

• MD 5004 bit 3	(NC START without reference point)
• MD 560* bit 4	(No reference point start disable)

If you have equipped axes with linear scales that have distance-coded reference marks, please note Section 11.16, Distance-coded reference marks.

11.3.2 Automatic identification of direction with approach to reference point

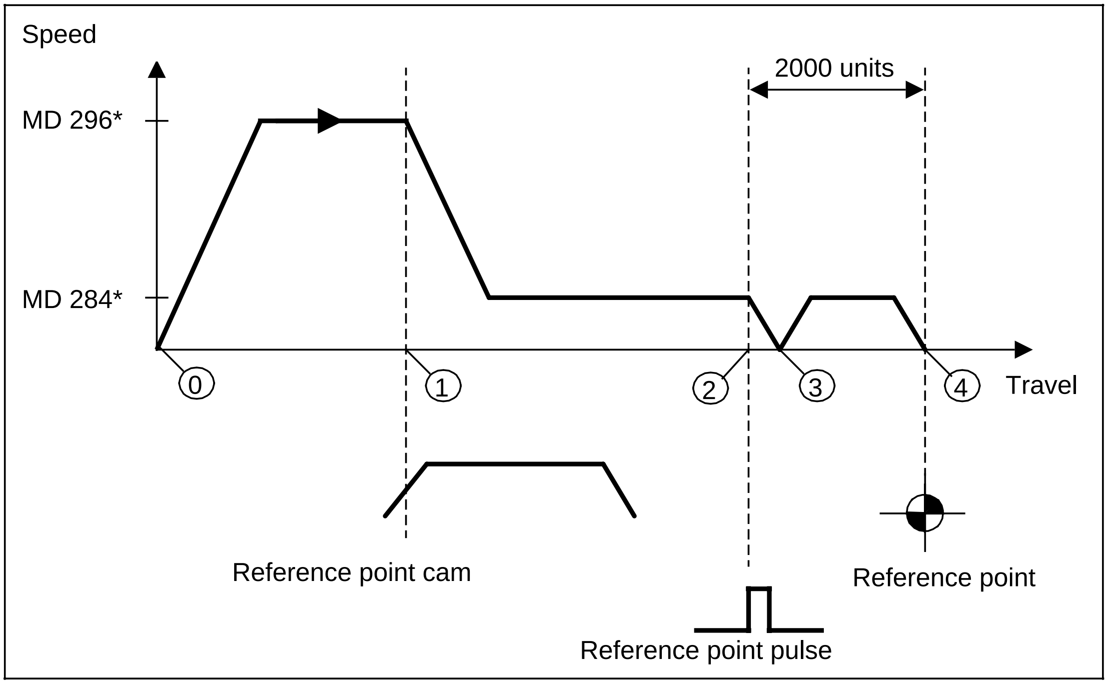
The control makes it possible to approach the reference point in two different ways, with or without automatic identification of direction. Selection is performed via MD 560* bit 6.

11.3.2.1 Approach to reference point without automatic identification of direction

Preconditions

- MD 560* bit 6 = 0
- Axis-specific feed enable set
- Common feed enable set
- Reference point between reference point cam and limit switch.

Case 1: Axis ahead of reference point cam



0 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*.

1 When the reference point cam is reached, the axis speed is reduced to the value in MD 284* via the "Deceleration" interface signal.

2 After the reference point cam has been left, the next reference point pulse is evaluated and the axis braked.

3 In order to prevent machine backlash during approach to the reference point, a 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.

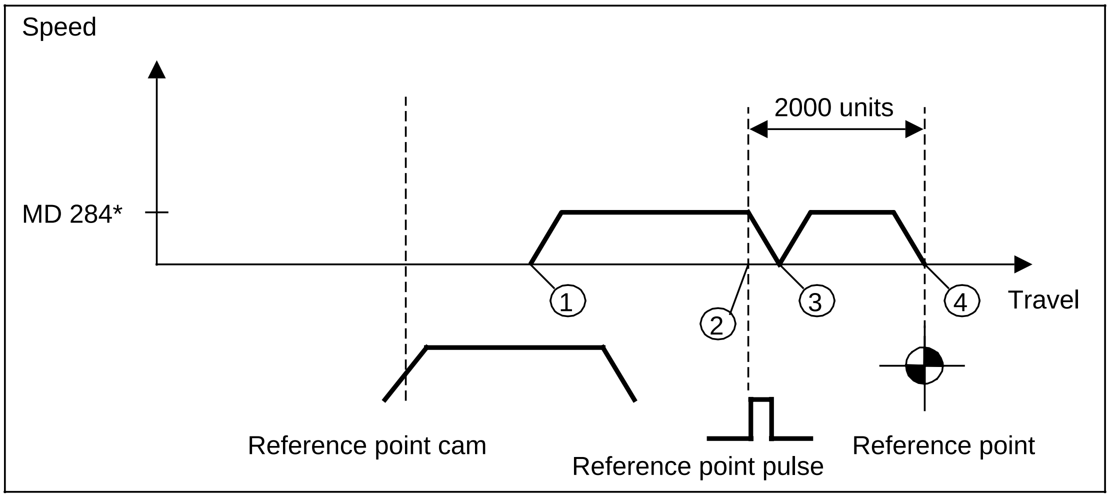
4 Reference point reached.

Note:

The distance travelled by the control (3) (4) after reaching the zero mark is dependent on the position control resolution entered in NC MD 5002. Thus after reaching the zero mark, the NC will travel another 2 mm given a specified position control resolution of $1/2 \times 10^{-3}$ mm but 20 mm when the position control resolution is $1/2 \times 10^{-2}$ mm. This distance could be compensated by means of NC MD 244* "Reference point shift".
Example: Input NC MD 244* = 2000 units of position control resolution $1/2 \times 10^{-3}$ mm. During approach to the reference point, the axis would move as far as (3) and then back to (2).

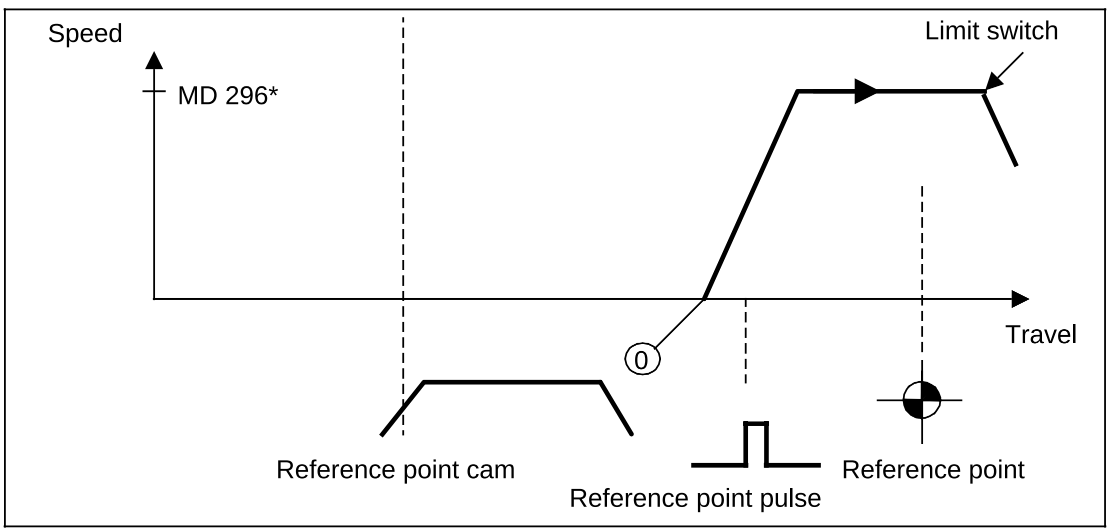
Case 2: Axis on reference point cam

The axis accelerates immediately to the reference cutoff speed (MD 284*) instead of the reference speed.



Case 3: Axis behind reference point cam

In view of the fact that 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 are not active prior to or during approach to the reference point.



Complex travel interlocks had to be integrated into the PLC to obviate Case 3. It was thus decided in the case of the SINUMERIK System 800 GA2 to offer a facility which follows on from 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.

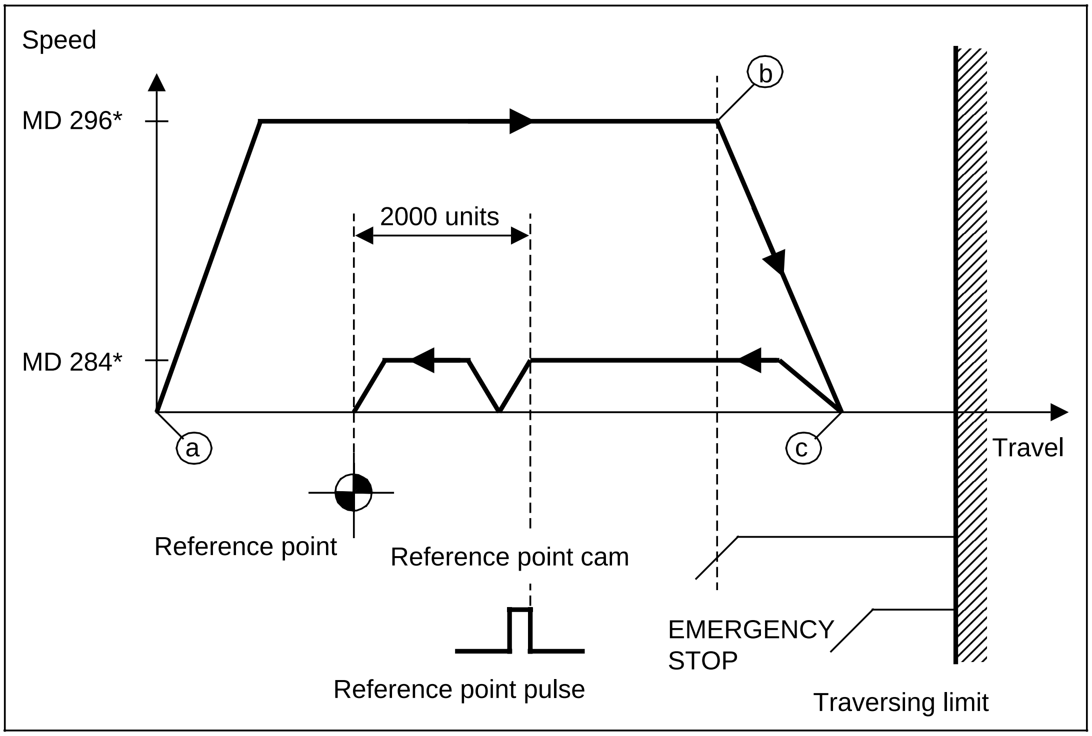
11.3.2.2 Approach to reference point with automatic identification of direction

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

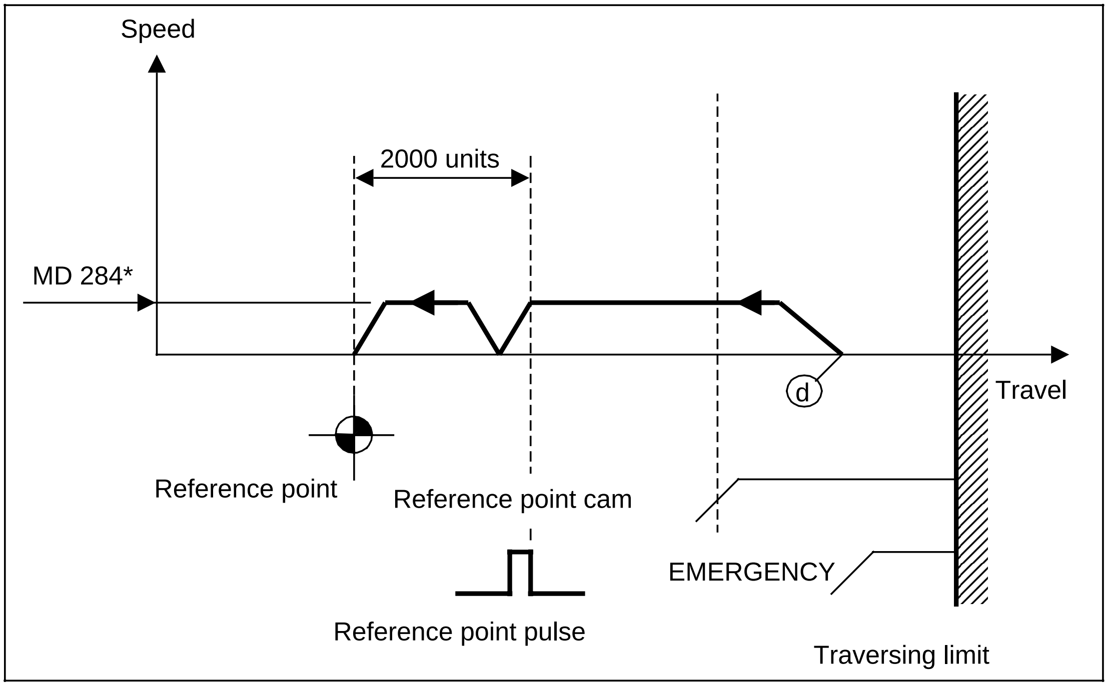


a When the direction key is actuated, approach to the reference point for the axis is initiated in the specified direction (MD 564* bit 0) at the speed in MD 296*.

b When the reference point cam is reached, the axis is braked to zero speed with the "Deceleration" signal.

c Deceleration from the reference point cam is performed at reverse speed (MD 284*) and the next reference pulse is evaluated (see Section 11.3.2.1 for a detailed description of the remainder of the sequence).

Case 2: Axis on reference point cam



ⓓ When the direction key is actuated, the control 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 11.3.2.1 for a detailed description of the remainder of the sequence).

Case 3: Axis behind reference point cam

This case cannot occur.

11.3.3 Reference point approach via program

The reference point can be approached with the programming function G74. The sequence is similar to that shown in Sections 11.3.2.1 or 11.3.2.2.

The defined direction (MD 564* bit 0) is started with G74 in the part program instead of with the direction key.

Example:

AUTOMATIC or MDI mode
N10 G74 X LF

Notes:

- Only one axis per NC block can be programmed (e.g. G74 C LF)
- TRANSMIT function and coupled motion must not be selected.
- G74 is active block by block.
- With G74 the tool offset and the zero offset PRESET + DRF are suppressed internally and become active again automatically after reference point reached. This also applies to G functions, such as G01, G93, G94 etc.
- After the function "Reference point approach with synchronization via program" has been started the "current position" is still updated. The "distance to go" is displayed as zero because during reference point approach there are no sensible values for distance to go.

11.4 Leadscrew error compensation

The LEADSCREW ERROR COMPENSATION function is an Option.

11.4.1 Corresponding data

- NC MD 316* (Pointer compensation +)
- NC MD 320* (Pointer compensation –)
- NC MD 324* (Distance between 2 leadscrew error compensation points)
- NC MD 328* (Compensation value)
- NC MD 6000 to 6999 (Leadscrew error compensation points)
- Option LEADSCREW ERROR COMPENSATION (H56) must be available

Caution!

Modification of all NC MD for leadscrew error compensation is not active until after POWER ON and reference point approach

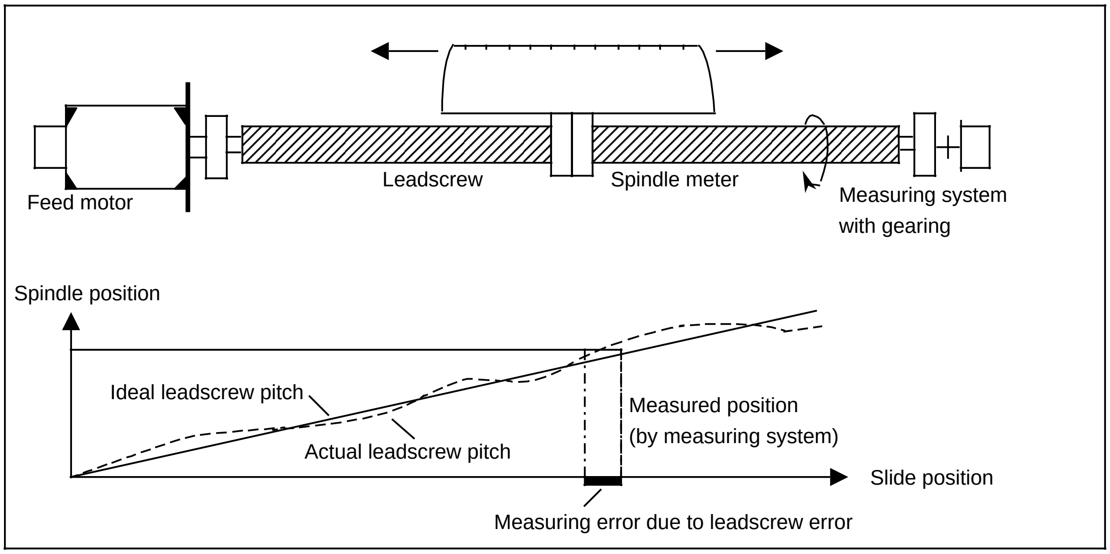
Direction-dependent leadscrew error compensation (in Option H56) must not be activated until leadsrew error compensation points (NC MD 6000 to 6999) have been entered and assigned to the error curve.

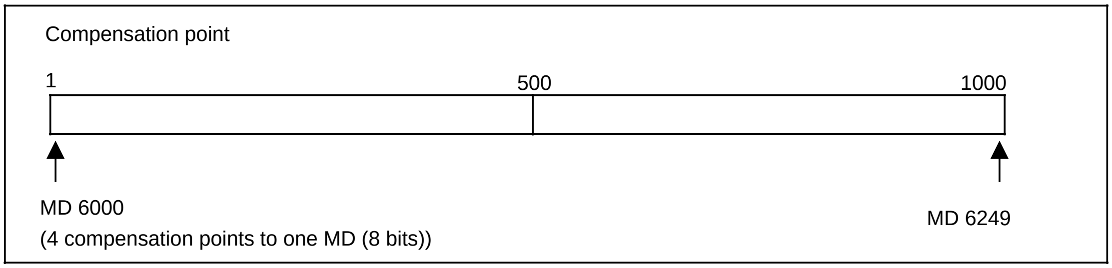
11.4.2 Function description

The principle of "indirect measurement" in NC 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.
The errors in all axes can be compensated separately. To this end, a total of 1000 compensation positions per servo CPU are available for all axes. The compensation position spacing for each axis is selectable over a range of 1 to 32.000 units. A compensating value of 0 to 100 units which is equal for all positions per axis may be set.

The leadscrew error compensation points lie in the machine data areas:

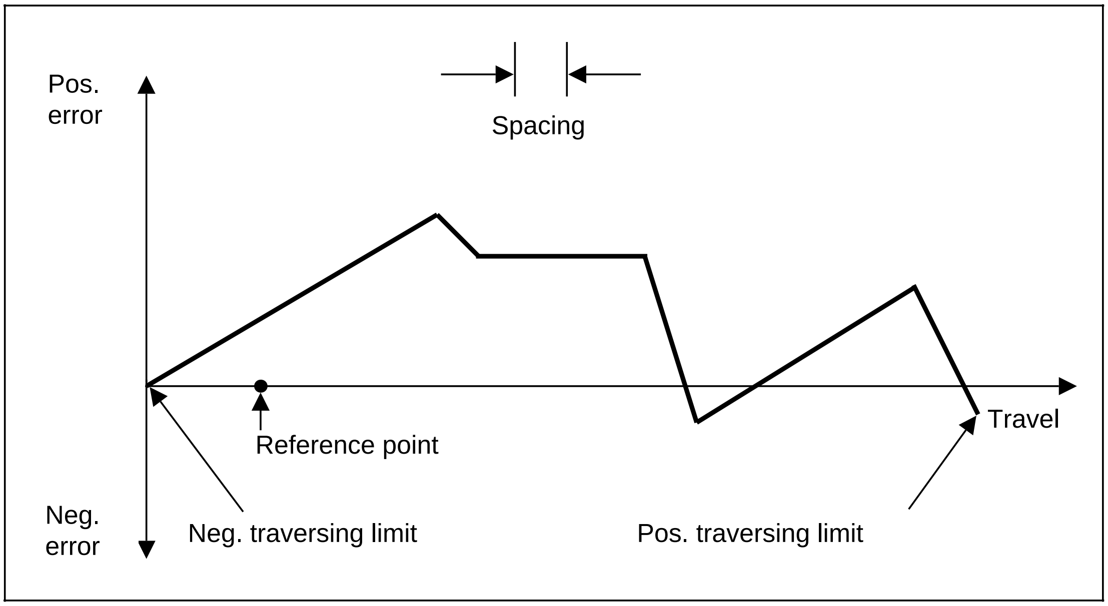
- MD 6000 to 6249 for axes of the 1st servo CPU
- MD 6250 to 6499 for axes of the 2nd servo CPU
- MD 6500 to 6749 for axes of the 3rd servo CPU
- MD 6750 to 6999 for axes of the 4th servo CPU



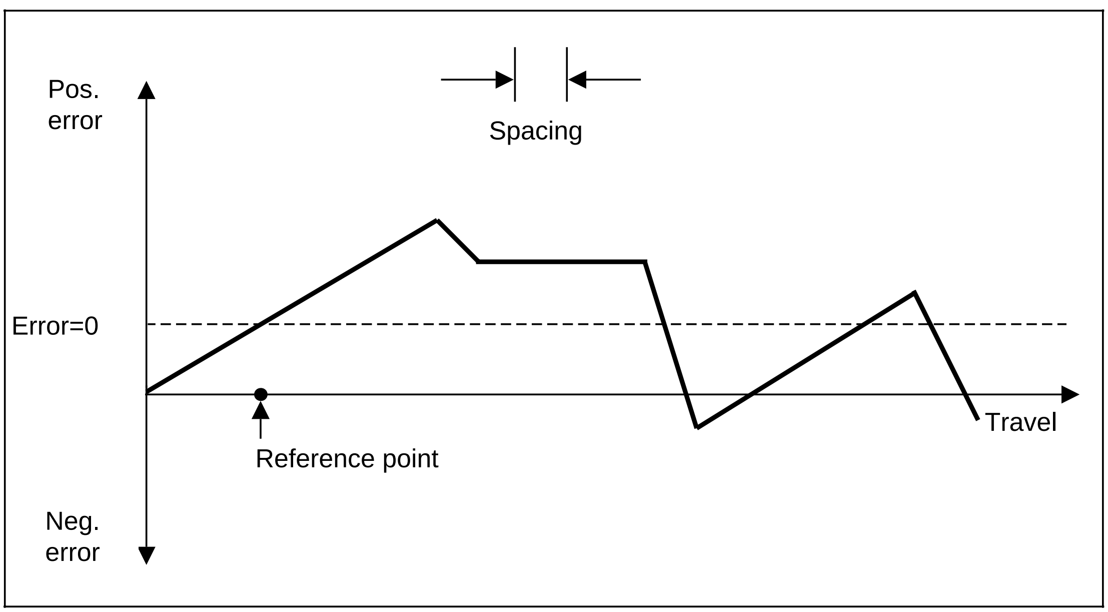


Measurement of leadscrew error compensation

The reference point is first approached to synchronize the measuring system. This is then followed by travel to the negative range limit of the axis, commencing from this point to plot an error curve in the positive direction using an accurate instrument; the reference point must be identified.



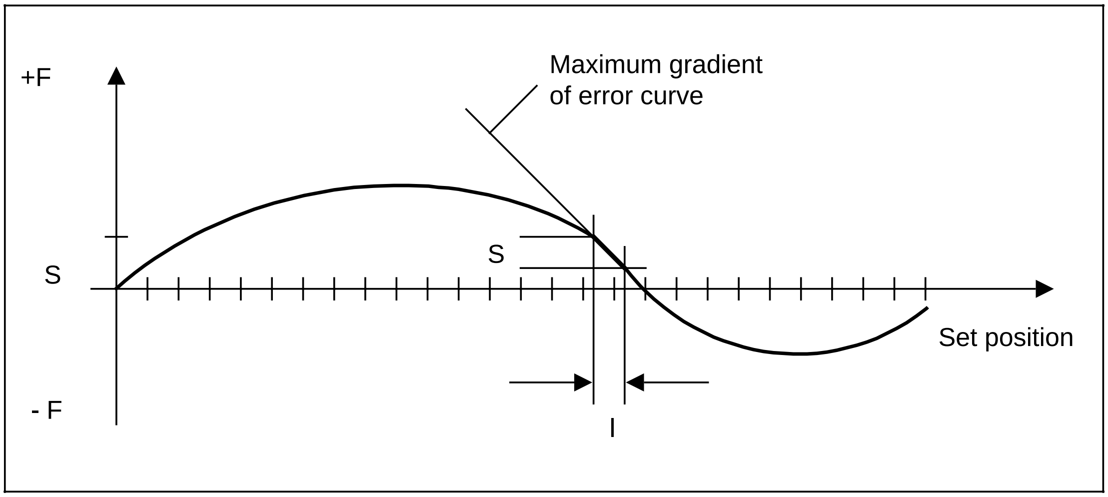
In view of the fact that compensation is not possible at the reference point, the error curve must be shifted so that the error is zero at the reference point.



The spacing between 2 leadscrew error compensation points (MD 324*) is then specified, being based on the permissible tolerance of the final (compensated) leadscrew error curve, the actual leadscrew pitch error and the number of possible compensating values.

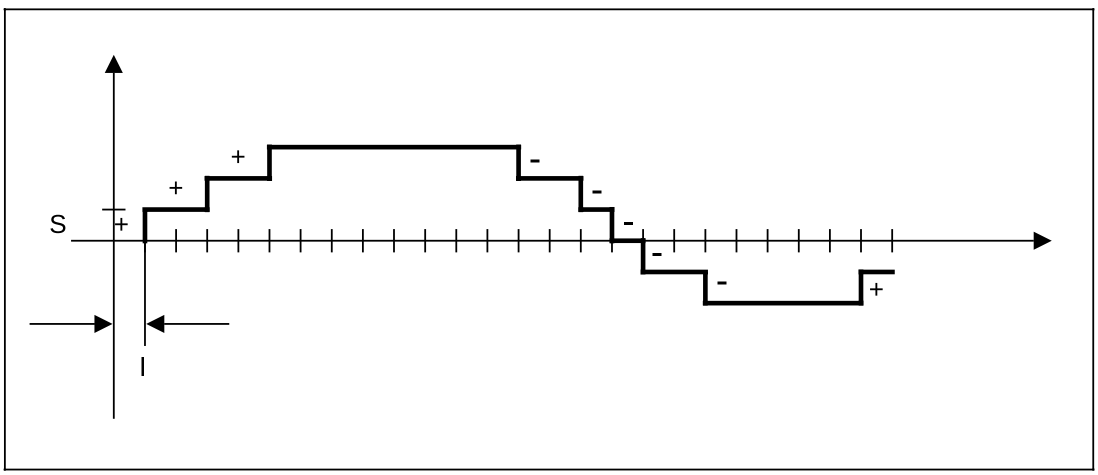
The following method may be used to determine the spacing between 2 leadscrew error compensation points:

S : Compensation amount, e.g. 1/2 tolerance band
I : Spacing between 2 leadscrew error compensation points

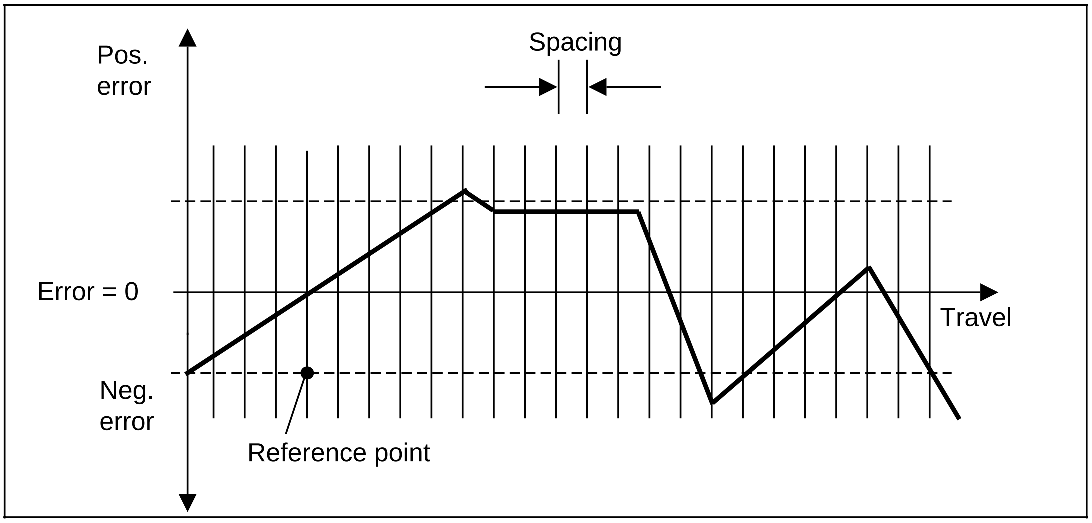


The point exhibiting the greatest error is determined together with the distance (I) covered by the specified compensation amount (S).

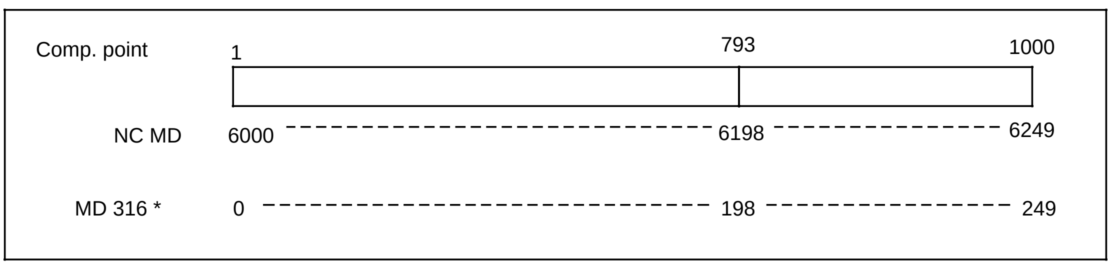
S	:	Compensation amount	= MD 328*
I	:	Spacing	= MD 324*



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. The compensating value (0 to 100 units) is transferred to NC MD 328*.



It is then specified how many compensating points must be supplied by way of the input 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.



In view of the fact that the SINUMERIK 880 GA2 has a total of between 1000 and 4000 compensation points for all axes, the controller must be informed by means of MD 316* as to which of the points corresponds to the axis reference point. The compensation point is not entered directly in MD 316*, MD offset (MD 6125 = MD offset = 125) for NC CPU 1 being entered instead, so the reference point can only be located on compensation points 1, 5, 9, 13, 17, ...

MD No.	Bit No.							
	7	6	5	4	3	2	1	0
6000	Comp. point 4 Yes / No + / -		Comp. point 3 Yes / No + / -		Comp. point 2 Yes / No + / -		Comp. point 1 Yes / No+ / -	
6001	Comp. point 8 Yes / No + / -		Comp. point 7 Yes / No + / -		Comp. point 6 Yes / No + / -		Comp. point 5 Yes / No + / -	
6002	Comp. point 12 Yes / No + / -		Comp. point 11 Yes / No + / -		Comp. point 10 Yes / No + / -		Comp. point 9 Yes / No + / -	
6248	Comp. point 996 Yes / No + / -		Comp. point 995 Yes / No + / -		Comp. point 994 Yes / No + / -		Comp. point 993 Yes / No + / -	
6249	Comp. point 1000 Yes / No + / -		Comp. point 999 Yes / No + / -		Comp. point 998 Yes / No + / -		Comp. point 997 Yes / No + / -	

- = 0
+ = 1
No = 0
Yes = 1

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:

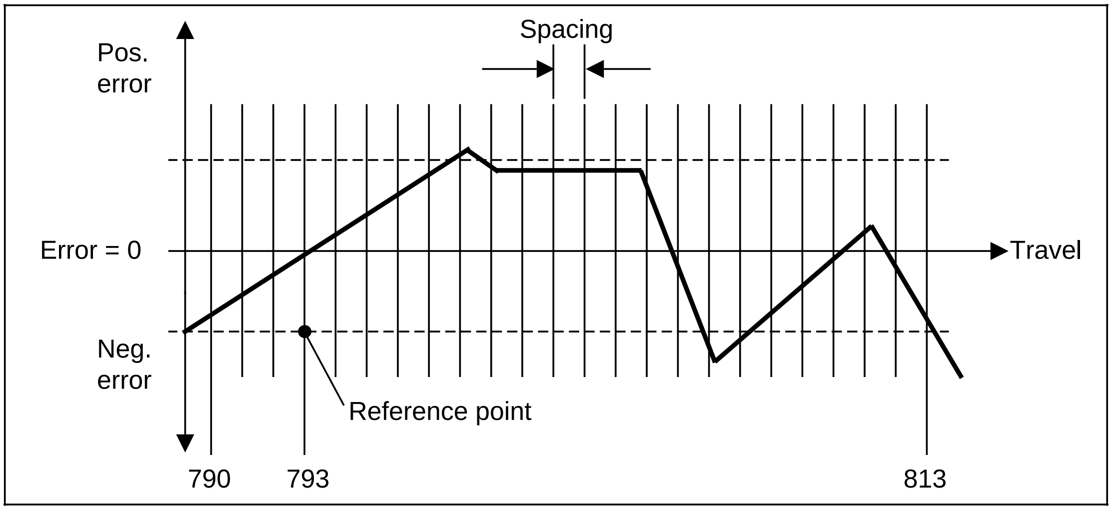
- Reference point to be at compensation point 793

$$\text{MD offset} = \frac{793 - 1}{4} = 198$$

- MD offset 198 = MD 6198

  Value in MD 316* ... 198

As stated above, the reference point determines which of the possible 1000 compensation points are used for the axis concerned. At this compensation point compensation is not possible.



Example:

Axis 1 exhibits the following error curve; no compensation points have been used so far.

Reference point value 0
Max. travel in negative direction - 35.000 mm
Max. travel in positive direction 205.00 mm

Tolerance band (specified by machine manufacturer), e.g. specify 0.01 mm spacing between 2 leadscrew error compensation points, e.g. 10 mm:

235 mm travel max.

A- **3 compensating values**

10 mm grid spacing

205 mm travel max.+

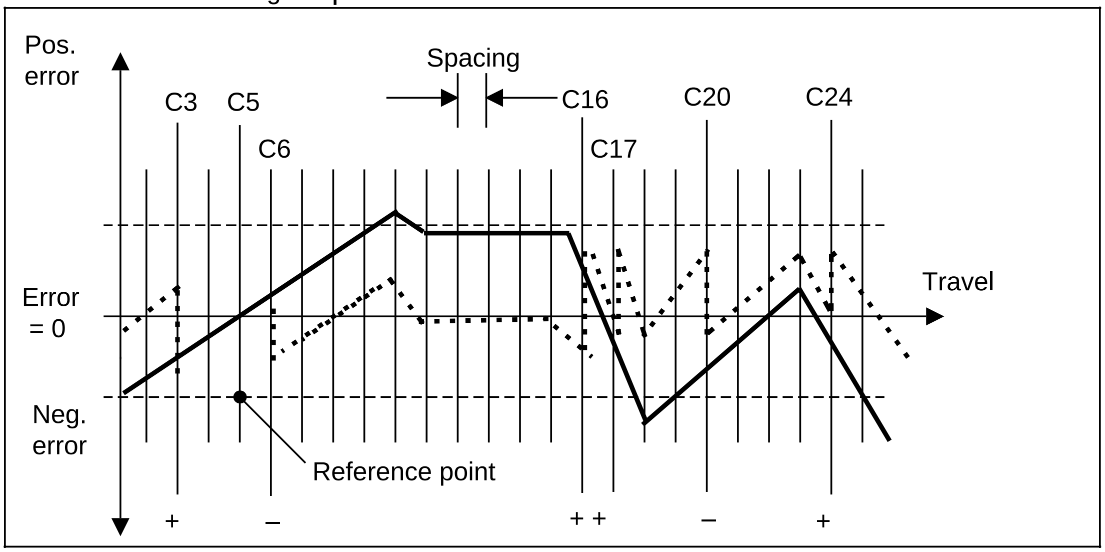
A+ **20 compensating values**

10 mm grid spacing

Total number of compensating values:
C =(A-) +(A+) + (Ref) = 3 + 20 + 1 **24 compensating values**

Thus for NC MD 3161 = 1 (NC MD 6001), i.e. compensation point C5 is the point at which the reference point is located; compensation may not be performed at this point. When 10 mm travel is performed in the negative direction, compensation point C4 is used for compensation. Compensation point C6 is used for 10 mm travel in the positive direction.

- - - Tolerance band e. g. 10 µm



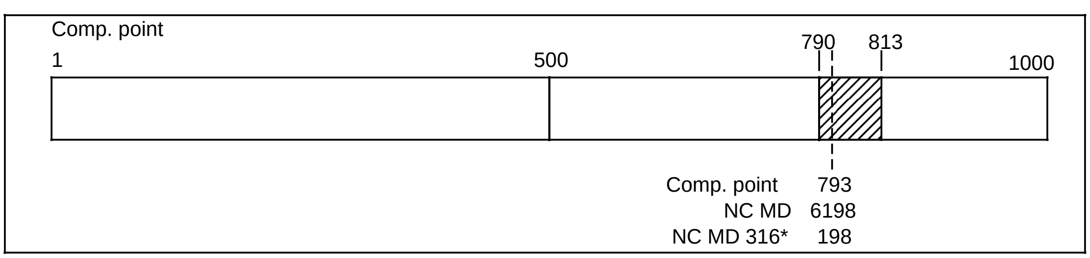
- - - - Compensating value, e.g. 5 µm
▪ ▪ ▪ ▪ ▪ Compensated curve

Starting from the reference point in the negative direction, the error curve extends as far as the traversing limit within the tolerance band. No compensation should be required. Better results are obtained if positive compensation is performed to C3.
In order to ensure that errors are maintained as near as possible to zero, compensation is required in the positive direction as follows: negative to C6, positive to C16 and C17, negative to C20 and positive once more to C24. The new error curve would then be as above.

The following machine data should be set:

- Leadscrew error compensation Option
- NC MD 3161 = 1 (specifying reference point, C5)
- NC MD 3241 = 10000 (grid spacing 10 mm)
- NC MD 3281 = 5 (compensating value 5 µm)
- NC MD 6000 = 00 11 00 00 (positive compensation, C3)
- NC MD 6001 = 00 00 10 00 (negative compensation, C6, bits 0 and 1 must be 0)
- NC MD 6002 = 0 No compensation
- NC MD 6003 = 0 No compensation
- NC MD 6004 = 00 00 11 11 (positive compensation, C15 and C16)
- NC MD 6005 = 00 00 10 00 (negative compensation, C20)
- NC MD 6006 = 00 00 11 00 (positive compensation, C24)

If the reference point is assigned to compensation point 793, breakdown of the 1000 compensation points is as follows:



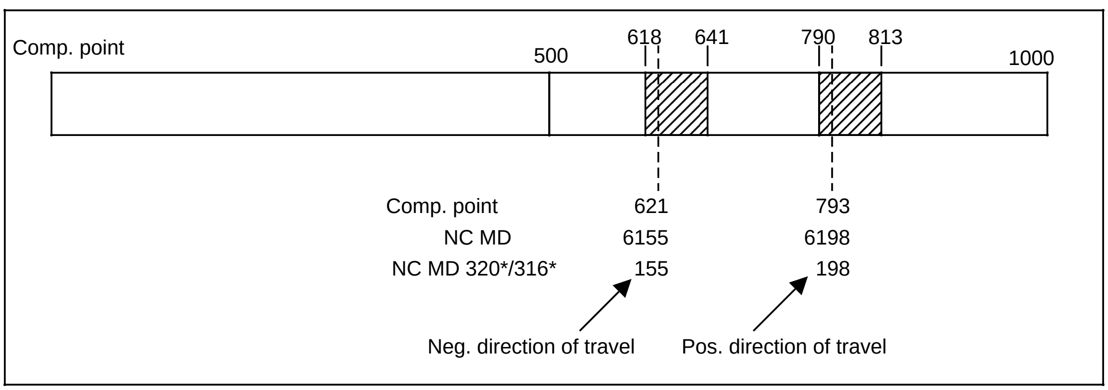
The reference point determines the location of the hatched area of the compensation points used. This area terminates at point 790 or 813 due to the spacing between the leadscrew error compensation points and the maximum traversing range of the axis.

If leadscrew error compensation is used for a number of axes, the commissioning engineer must ensure when inputting the MD 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. With direction-dependent leadscrew error compensation, there is - similar to leadscrew error compensation - a second compensation curve plotted from the positive to the negative direction.

In the case of ball screws, pre-stressing 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 entering the positive compensation values, 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:



Both direction-dependent and direction-independent leadscrew error compensations are Options and must therefore be ordered (H 56).

Any modification to the MD 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 compensation value (NC MD 328*) is limited to max. 100 units.

11.5 Rotary axis functions

11.5.1 Corresponding data

All data as with linear axes, but with the following additions/supplements:

- NC MD 344* (Modulo value rotary axis for leadscrew error compensation)
- NC MD 560* bit 7 (Actual value display modulo 360°)
- ND MD 560* bit 3 (Rounding values with rotary axes)
- NC MD 560* bit 2 (Rounding to whole/half degrees)
- 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 (Traverse rotary axis modulo 360°)
- NC MD 5002 (Input resolution)
- NC MD 1800* (Position control resolution)
- Alarm 100* (Spindle lead grid distance invalid)
- Alarm 2064 (Program error rounding axis)

11.5.2 Function description

A rotary axis has to meet different requirements, depending on the machine type. The rotary axis function is therefore subdivided into three part functions which are activated by means of the machine data or program.

Combining the part functions allows the control to be matched to the various machine types.

Graphic simulation of rotary axes is not possible.

”Rotary axis”: NC MD 564* bit 5

This machine data defines the axis as a rotary axis. The display is absolute (1 revolution 360°, 2 revolutions 720° etc.), as are the @ functions. However, the axis is programmed like a linear axis. The axis-specific NC MD units are treated otherwise.

Unit 10^{-3} degrees given a position control resolution of 1/2 x 10^{-3} units and an input resolution of 10^{-3} units.

Actual-value display ”Modulo 360°”: NC MD 560* bit 7

"Modulo" display when the bit is set, i.e. after 359.999 degrees the actual-value display is reset to 0. The axis is programmed in the same way as a linear axis.

”Modulo programming”: NC MD 572* bit 2

After powering up and after block search, the shortest path is always processed with the machine data set. The G68 function allows the block end value always to be approached over the shortest path.

G68 is modal and belongs to the G90/91 group. G68 is treated in the same way as G90 if "modulo programming" is not activated.

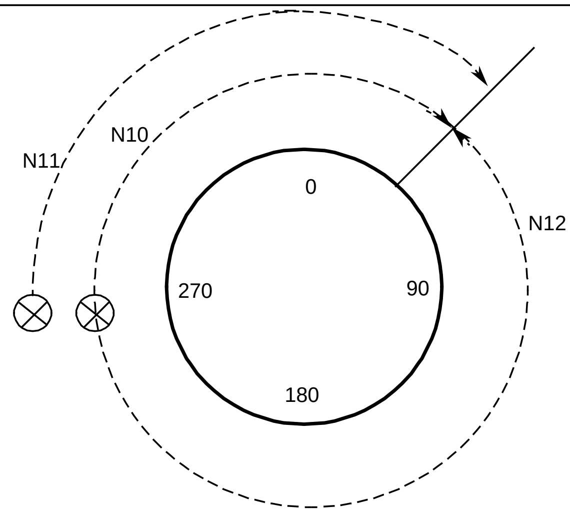
If the rotary axis is not to cover the shorter path, it must be programmed with G90 and a sign. If a "modulo" rotary axis (NC MD 572* bit 2 = 1) is switched to follow-up mode, NC Start automatically activates G58 for the next traversing block of this rotary axis.

Example: Axis at 270°

N10 G68 C10

N11 G90 C10

N12 G90 C-0



Machine data "Modulo 360°" and "Modulo programming" are only allowed with the "Rotary axis" machine data.

"Traverse rotary axis modulo 360°": NC MD 572* bit 4

After zero or tool offsets have been taken into account and in conjunction with the "delete distance to go" function, the internal data adjustments can lead to traversing movements of greater than ±360 (180) degrees with G90 (G68). This can be avoided by setting NC MD 572* bit 4. This ensures that a G90 (G68) block results in a max. movement of ±360 (180) degrees.

Combination of part functions:

Rotary axis	Modulo program	Modulo 360°	Remarks
0	0	0	Linear axis
1	0	0	Application permitted
1	0	1	
1	1	1	
1	1	0	Application prohibited
0	0	1	
0	1	1	
0	1	0	

The Function Manual "Extended Spindle Functions" contains information on using spindles as rotary axes (C axis operation).

Axis-specific machine data

If an axis is defined as a rotary axis (NC MD 564* bit 5), NC MD 5002 (input resolution) and NC MD 1800* (position control resolution and display resolution) must be stated in the same unit system.

11.6 Warm restart

11.6.1 Corresponding data

- NC MD 100* (Mode group number)
- NC MD 360* (Axis valid in mode group)
- NC MD 453* (Spindle valid in mode group)
- Signal DB 48 DL 0 bit 0 (Initiate warm restart)
- Signal DB 48 DR 1 bit 0 (Warm restart ended)
- Alarm No. 70 to 80
- Alarm No. 3023
- Alarm No. 3014

11.6.2 Function description

In some machines, arbitrary axes must be assigned to another mode group (BAG) without having to shut down the control since the reference point is then lost.

Example:

A machine has 2 working areas:
Working area I with axis B'
Working area II with axes B1'

The working areas are broken down into different operating groups to permit efficient working. The three main axes X, Y, Z are then assigned separately via the warm restart function to working area I (mode group 1) or working area II (mode group 2). Each working area requires on indendent machine control panel to which main axes X, Y, Z are assigned in each case.

((Bild aus: E80850-D1-X-A3 / Seite 11-32
(Arbeits-Nr. 886154)
1:1 einmontieren))

If work is performed in working area I, the three main axes X, Y, Z, axis B' and the spindle are assigned to machine control panel 1 or mode group 1.

In working area II, axes B1' can be moved in jog mode to load and unload a workpiece.

If a workpiece is now to be machined in working area II with clamping and unclamping in working area I, the axis assignments must be modified, i.e. the mode groups must be switched. During this changeover, the axes are re-assigned to the machine control panels.

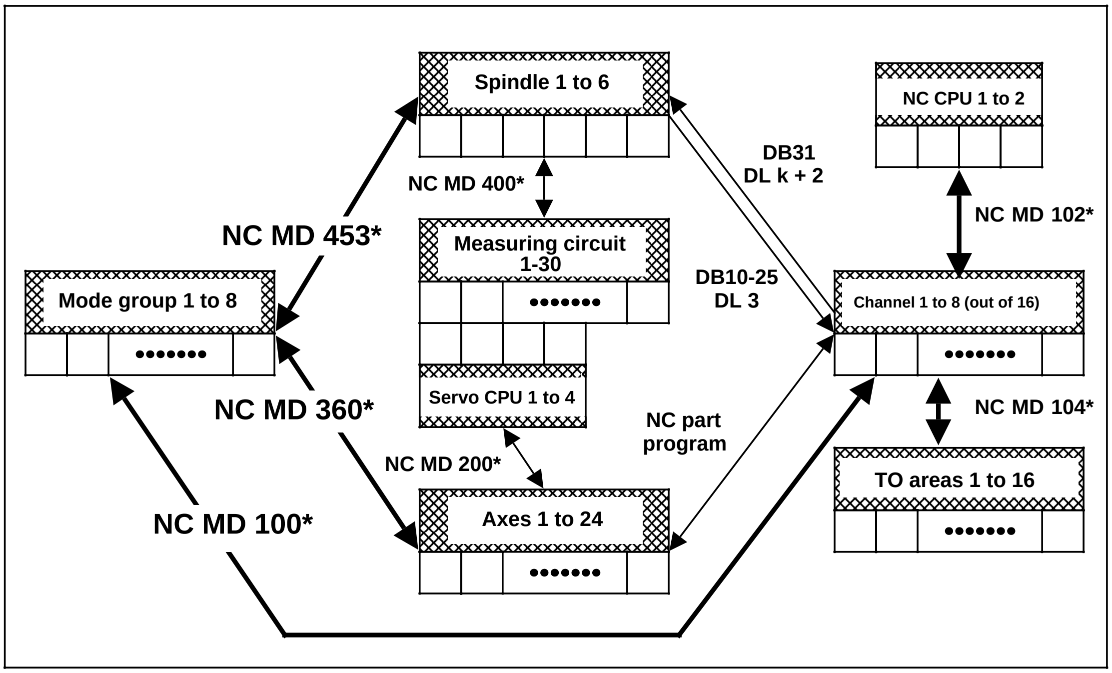
An auxiliary function is assigned to each working area for mode group switching purposes. When the respective auxiliary function is output, the PLC recognizes the necessary axis configuration and corrects it if necessary in the machine data. Once the machine data have been modified, the "warm restart" function must be activated at the interface by the PLC (DB48 DL0 bit 0). This function reconfigures the controller without the reference points having to be approached once more.

The machine data may also be modified using @ functions by calling a cycle from the PLC in an unassigned channel.

Warm restart must be performed when the following machine data are modified:

• NC MD 100*	Channel valid in mode group
• NC MD 316*	Pointer for leadscrew error compensation
• NC MD 320*	Pointer for leadscrew error compensation
• NC MD 360*	Axis valid in mode group
• NC MD 453*	Spindle valid in mode group
• NC MD 876 to 899	Coupled-axis grouping
• NC MD 5156 to 5183	Coupled-motion combination
• NC MD 5060 to 5139	Transformation data blocks

If a different reconfiguring machine data (active: after POWER ON) has been modified, no warm restart can be executed thereafter, but a hardware Reset (POWER ON) has to be performed instead.



The following MD do not necessitate a warm restart when modified:

NC MD		Modification accepted
104*	TO range number	immediately
304*	IPO parameter name	after each NC block
548*	Axis name, horizontal axis	after each NC block
550*	Axis name, perpendicular axis	after each NC block
552*	Axis name, vertical axis	after each NC block
554*	Axis with constant cutting speed	after each NC block
576*	Axis not allowed in channel	after each NC block
580*	Axis not allowed in channel	after each NC block

Warm restart may only be performed with all channels in the RESET state, i.e. all channels must be put into the RESET state by the PLC, coupled motion and transformation must be deselected and all spindles stopped (with M19 active) before the warm restart is initiated. During the warm restart, the RESET status in all NC channels must not be cancelled. The axial, spindle-specific interface as well as the "EMERGENCY STOP" signal at the NC PLC and PLC NC interfaces is not processed during warm restart (control in reset state).

Warm restart functional sequence

1. NC
   - NC MD 360* is defined using CL 800 instructions (also feasible via PLC)
   - Auxiliary function for warm restart request is output, e.g. H1234
2. PLC
   - Auxiliary function is interpreted
   - NC MD 360* is defined by PLC (also feasible via NC)
   - Reset via PLC for all channels
3. NC
   - **all** part programs are aborted (channels 1 to 16)

In steps 1, 2 or 3 all spindles must be stopped.

4. PLC
   - User sets at interface: "warm restart" 1 (DB 48 DL 0 bit 0)
5. NC
   - Sets interface signal: "warm restart terminated" 1 (DB 48 DR 2 bit 0)
6. PLC
   - User clears interface: "warm restart" 0
7. NC
   - Clears interface "warm restart terminated" 0
8. Warm restart function terminated, program operation can be resumed.

Fault response

If machine data errors are established during warm restart, the NC activates Alarms 70 to 80. In this case, the acknowledgement signal "warm restart terminated" is set. The alarm may only be cleared by means of POWER ON after correcting the appropriate machine data (see also description of alarms, Section 12).

11.7 Axis-specific resolutions

11.7.1 Corresponding data

MD 5002	bit 4 - 7	Input resolution
MD 564*	bit 5	Rotary axis
MD 1800*	bit 0 - 3	Axis-specific position control resolution
MD 1800*	bit 4 - 7	Axis-specific display resolution

Indirectly related:

MD 155	Interpolation time
MD 160 - 163	Ratio of interpolation to position control

11.7.2 General remarks on the axis-specific resolutions

The axes can be matched to the machine via NC MD.
It must be remembered that only specific combinations are permissible, and care must be taken that the boundaries, maximum axis speed and range limits are not exceeded.

The following types of resolution can be specified for axes:

- Input resolution: Set via MD for all axes
- Geometry resolution: Input resolution x 0.5
- Position control resolution: Set via MD for each axis
- Display resolution: Set via MD for each axis
- Measuring system resolution: Set by the measuring system for each axis

The measuring system resolution is adapted to the position control resolution using the variable increment weighting (NC MD 364* and 368*).

11.7.3 Input, display and position control resolution

Input resolution (Input System - IS)

The input resolution for the entire control is defined in MD 5002, bits 4 to 7. The input resolution defines the geometry resolution for linear and rotary axes. Rotary axes have the same input resolution as linear axes. The geometry resolution determines the interpolation accuracy.

With the input resolution the maximum number of programmable decimal places for position values in the part program and the number of decimal places for TO, ZO, SD etc. are also defined (and therefore the maximum attainable accuracy).

The input resolution defines the units system (inch - metric - degrees).

The input resolution must be taken into account when entering machine data that must be stored in the input system. The default value for linear axes is 10^{-3} mm and for rotary axes 10^{-3} degrees.

Display resolution

In addition to the input resolution, the user must also define the display resolution. In contrast to the input resolution, the display resolution is defined separately for each axis. NC MD 1800* bits 4-7, are provided for this purpose. The display resolution defines the number of decimal places of the actual position and distance to go that are displayed on the NC screen. The default value for all axes is 10^{-3} mm or degrees.

The display resolution must have the same input system (inch - metric - degrees) as the input resolution. Alarm 4 (Power on Alarm) is issued to flag "Illegal input system" if this is not the case.

Position control resolution (Measuring System - MS)

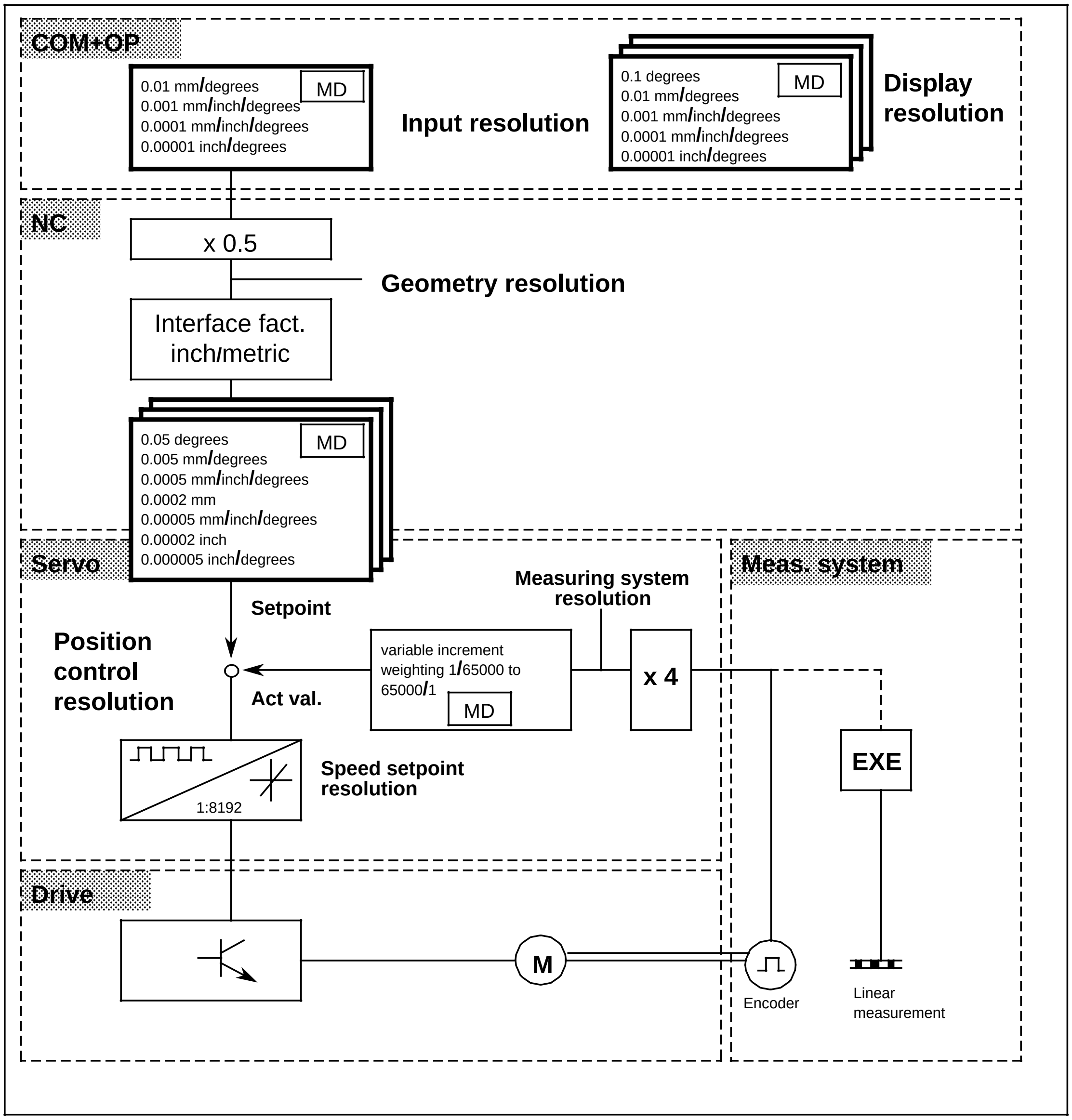
Like the display resolution, the position control resolution is defined on an axis-specific basis. This must be taken into account when entering machine data stored in the measuring system. MD 1800* bits 0 to 3, are used to define the position control resolution.

The unit system must be the same for each position control resolution.
The unit system (inch - metric - degrees) used for the position control resolution need not necessarily be identical to the one used for the input resolution.

Note:

A position control resolution $<10^{-3}$ degrees is only possible for rotary axes if Option E50 is used.

11.7.4 Block diagram of the resolutions



MD .... Machine data

11.7.5 Resolution codes

The following table shows the codes for the various types of resolution.
Alarm 4 ("Illegal input system") is issued when illegal values are entered as machine data.
MD 5002 bit 4 is used as the unit system identifier. Metric input system G71 (bit 4 = 0) is the reset state.

Table of resolution codes

Bit 7	Bit 6	Bit 5	Bit 4	Input resolution			NC MD 5002
Bit 7	Bit 6	Bit 5	Bit 4		Display resolution		NC MD 1800*
Bit 3	Bit 2	Bit 1	Bit 0			Pos. contr. resol.	NC MD 1800*
0	0	0	0	————	10^{-1} [mm] [degr.]	0.5 x10^{-1} [mm] [degr.]	Metric (degrees)
1	0	0	0	10^{-2} [mm] [degr.]	10^{-2} [mm] [degr.]	0.5 x 10^{-2} [mm] [degr.]	
0	1	0	0	10^{-3} [mm] [degr.]	10^{-3} [mm] [degr.]	0.5 x 10^{-3} [mm] [degr.]	
1	1	0	0	————	————	2 x 10^{-4} [mm] [degr.]	
0	0	1	0	10^{-4} [mm] [degr.]	10^{-4} [mm] [degr.]	0.5 x 10^{-4} [mm] [degr.]	
1	0	1	0	10^{-5} [mm] [degr.]	10^{-5} [mm] [degr.]	0.5 x 10^{-5} [mm] [degr.]	
0	1	1	0	————	————	————	
1	1	1	0	————	————	————	
0	0	0	1	————	10^{-1} [degr.]	0.5 x 10^{-1} [degr.]	Inch (degrees)
1	0	0	1	————	10^{-2} [degr.]	0.5 x 10^{-2} [degr.]	
0	1	0	1	10^{-3} [inch] [degr.]	10^{-3} [inch] [degr.]	0.5 x 10^{-3} [inch] [degr.]	
1	1	0	1	10^{-4} [inch] [degr.]	10^{-4} [inch] [degr.]	0.5 x 10^{-4} [inch] [degr.]	
0	0	1	1	————	————	2 x 10^{-5} [inch]	
1	0	1	1	10^{-5} [inch] [degr.]	10^{-5} [inch] [degr.]	0.5 x 10^{-5} [inch] [degr.]	
0	1	1	1	10^{-6} [inch] [degr.]	————	————	
1	1	1	1	————	————	————	

0 1 0 0 = standard machine data

11.7.6 Permissible combinations of resolutions

Input resolution, display resolution and position control resolution can be freely defined within certain limits (see the following two tables). Note the between the input resolution and the position control resolution for all axes together a factor of up to 200 is possible.

Example:		
	Input resolution	10^{-4} mm or degrees
	Position control resolution for rotary axis	$0.5 \cdot 10^{-4}$ degrees
	Position control resolution for machining axes	$0.5 \cdot 10^{-3}$ mm
	Position control resolution for loader axes	$0.5 \cdot 10^{-2}$ mm

Valid combinations of position control resolution and input resolution

Unit system	Position control resolution	Input resolution							
		10^{-2} [mm] [degr.]	10^{-3} [mm] [degr.]	10^{-4} [mm] [degr.]	10^{-5} [mm] [degr.]	10^{-3} [inch] [degr.]	10^{-4} [inch] [degr.]	10^{-5} [inch] [degr.]	10^{-6} [inch] [degr.]
mm	0.5×10^{-1} [degr.]	y	y	-	-	-	-	-	-
mm	0.5×10^{-2} [mm][degr.]	xy	xy	xy	-	x	x	x	-
mm	0.5×10^{-3} [mm][degr.]	xy	xy 1)	xy	-	x	x	x	-
mm	2×10^{-4} [mm]	x	x	x	x	-	-	x	x
mm	0.5×10^{-4} [mm][degr.]	xy	xy	xy	xy	-	-	x	x
mm	0.5×10^{-5} [degr.]	y	y	y	y	-	-	-	-
inch	0.5×10^{-1} [degr.]	-	-	-	-	y	-	-	-
inch	0.5×10^{-2} [degr.]	-	-	-	-	y	y	-	-
inch	0.5×10^{-3} [inch][degr.]	x	x	-	-	xy	xy	x	-
inch	0.5×10^{-4} [inch][degr.]	x	x	-	-	xy	xy	xy	-
inch	2×10^{-5} [inch]	x	x	x	x	x	x	x	x
inch	0.5×10^{-5} [inch][degr.]	x	x	x	x	xy	xy	xy	xy

x ... Linear axes only
y ... Rotary axes only
xy ... Linear and rotary axes
– ... Linear axes and rotary axes not permitted
1) ... Standard machine data

Permissible combinations of position control resolution and display resolution

Unit system	Position control resolution	Display resolution									
		mm					inch				
		10^{-1}	10^{-2}	10^{-3}	10^{-4}	10^{-5}	10^{-1}	10^{-2}	10^{-3}	10^{-4}	10^{-5}
mm	0.5×10^{-1} [degr.]	xy	-	-	-	-	-	-	-	-	-
mm	0.5×10^{-2} [mm][degr.]	-	xy	-	-	-	-	-	x	-	-
mm	0.5×10^{-3} [mm][degr.]	-	-	xy	-	-	-	-	-	x	-
mm	2×10^{-4} [mm]	-	-	xy	xy	-	-	-	-	-	x
mm	0.5×10^{-4} [mm][degr.]	-	-	-	xy	-	-	-	-	-	x
mm	0.5×10^{-5} [degr.]	-	-	-	-	xy	-	-	-	-	-
inch	0.5×10^{-1} [degr.]	-	-	-	-	-	xy	-	-	-	-
inch	0.5×10^{-2} [degr.]	-	-	-	-	-	-	xy	-	-	-
inch	0.5×10^{-3} [inch][degr.]	-	x	-	-	-	-	-	xy	-	-
inch	0.5×10^{-4} [inch][degr.]	-	x	-	-	-	-	-	-	xy	-
inch	2×10^{-5} [inch]	-	-	x	-	-	-	-	-	xy	xy
inch	0.5×10^{-5} [inch][degr.]	-	-	x	-	-	-	-	-	-	xy

xy ... Permissible for both linear and rotary axes
x ... Permissible for linear axes only
– ... Linear axes and rotary axes not permitted

Caution:

The display resolution must have the same system (inch-metric-degree) as the input resolution.

11.7.7 The influence of resolution on speed

The input resolution determines not only the path resolution, but also the lowest programmable speed. The lowest programmable speed is always 10 times coarser than the path resolution. For example, if the input resolution is 10^{-4} mm, the lowest programmable speed is then 10^{-3} mm/min. Depending on the unit system used, the feedrate is interpreted either in mm/min or in inches/min. If one of the interpolating axes is a rotary axis, the corresponding axial feedrate is interpreted as degrees/min.

When rotational feedrate G95 is used, the feedrate is interpreted in either mm/revolution, inches/revolution or degrees/revolution. When the feedrate is rotational, the lowest programmable speed is identical to the path resolution (for instance, for an input resolution of 10^{-4}, the lowest programmable speed for G95 would be 10^{-4} mm/revolution).

Input resolution	Min. programmed path speed
10^{-2} mm, degr.	0.1 mm/min, degr./min
10^{-3} mm, degr.	0.01 mm/min, degr./min
10^{-4} mm, degr.	0.001 mm/min, degr./min
10^{-5} mm, degr.	0.0001 mm/min, degr./min
10^{-3} inch, degr.	0.01 inch/min, degr./min
10^{-4} inch, degr.	0.001 inch/min, degr./min
10^{-5} inch, degr.	0.0001 inch/min, degr./min
10^{-6} inch, degr.	0.00001 inch/min, degr./min

With the SINUMERIK 880 GA2 the maximum axis speed depends only on the input resolution:

Input resolution	Max. axis speed (NC MD 540*.6 = 0 mm/min)	Max. axis speed (NC MD 540*.6 = 1 m/min)
10^{-2} mm	1072 0000 mm/min	2144 0000 mm/min
10^{-3} mm	1072 0000 mm/min	2144 0000 mm/min
10^{-4} mm	107 2000 mm/min	2144 0000 mm/min
10^{-5} mm	10 7200 mm/min	2144 0000 mm/min
10^{-3} inch	10 7200 inch/min	
10^{-4} inch	10 7200 inch/min	
10^{-5} inch	10 7200 inch/min	
10^{-6} inch	1 0720 inch/min	

The maximum path speed (defined by the input resolution) and the maximum axis speed together define the maximum speeds.

The interpolator breaks the tool path down into the associated axis-specific speed components (axis speeds). These values are then converted into position control resolution. Conversion is possible, however, only when the relevant maximum speeds and their correlation to interpolation are observed during programming. Alarm 2038, i.e. "Tool path too high" (Reset alarm) is issued if the speed is not possible. The alarm inhibits processing and NC start.

Setting data item "Dry run feedrate" is still entered in 1000 IS units/min. As many as 5 digits can be displayed and entered via keyboard. (For example, input resolution 10^{-4} mm dry run feedrate 9.999 m/min).

Alarm 3040 ("Field/variable cannot be displayed") is issued if more than 5 digits are programmed via the CL 800 command "SEN" or the @ 410 command.

11.7.8 Maximum speed for thread cutting

In the case of threading blocks G33, G34, G35 and G37 the feedrate is computed from the spindle speed and the pitch rather than being based on the programmed (linear) feedrate. This feedrate determines the tool path feedrate for the threading block.

Constant-pitch thread cutting (G33, G37)

For this type of threading, the tool path feedrate may not exceed the following limiting values.

Input resolution:	10^{-2} mm	:	<	1000	m/min
	10^{-3} mm	:	<	1000	m/min
	10^{-4} mm	:	<	1000	m/min
	10^{-5} mm	:	<	100	m/min

In addition to the maximum tool path feedrate, the maximum axis speed must also be taken into account when programming threading blocks (see tables in Section 11.7.7).

Variable-pitch thread cutting (G34/G35)

The valid limiting values must be observed for this type of thread. This applies to tool path feedrate, axis speed, pitch and spindle speed (see Programming Guide).

11.7.9 Maximum traversing range

The set combination of input resolution and axis-specific position control resolution determines the max. traversing range (for every axis separately). This maximum traversing range applies to the max. path between the two axis movements and for the max. programmable value for axis positions, interpolation parameters, chamfer, radius, etc.

The values defined as working area limitation (setting data 300*, 304*) must lie within the area boundaries. Alarm 3084 ("Invalid working area limitation") is issued if one of these values is not in range. In addition, the control enters the highest or lowest permissible value in SD "working area limitation".

In addition to the working area limitation, the software limit switches (MD 224*, 228*, 232*, 236*) and the pre-limit switches (MD 1100*) are also checked for validity. Alarm 87 ("Invalid software limit switch") is issued if impermissible values are detected. The validity check is carried out without regard to MD 556* bit 5 ("Working area limitation, software limit switches in force").

In order to prevent the working area from being exceeded, the absolute position is compared with the limiting values. This comparison is necessary only when the software limit switches/working area limitation are not active, as exiting of the traversing range is otherwise impossible. This check is not carried out for rotary axes.

Alarm 1204* ("Traversing limit") (Reset alarm) is issued when a limit violation is detected. The alarm is issued for limit violations in both directions of travel. The same reaction is initiated when the working field bounds are violated, i.e. withdrawal is possible in the reverse direction only. The alarm is not issued until the traversing limit is exceeded. The relevant axes are then abruptly braked (no deceleration ramp).

In the case of endlessly revolving rotary axes, the traversing range can be legally exceeded if a permissible combination of input resolution and position control resolution was defined when configuring the system. Impermissible combinations of input resolution and position control resolution will result in a malfunction (when the traversing limit is exceeded).

Unit system	Position control resolution	Input resolution 10^{-2} [mm] [degr.]	10^{-3} [mm] [degr.]	10^{-4} [mm] [degr.]	10^{-5} [mm] [degr.]
inch	0.5*10^{-1} [degr.]	-- -- --	-- -- --	-- -- --	-- -- --
inch	0.5*10^{-2} [degr.]	-- -- --	-- -- --	-- -- --	-- -- --
inch	0.5*10^{-3} [inch] [degr.]	±99999.99 mm ±3937.007 inch --	±99999.999 mm ±3937.0078 inch --	-- -- --	-- -- --
inch	0.5*10^{-4} [inch] [degr.]	±99999.99 mm ±393.700 inch --	±9999.999 mm ±393.0078 inch --	-- -- --	-- -- --
inch	2*10^{-5} [inch]	±25399.99 mm ±999.999 inch --	±25399.999 mm ±999.9999 inch --	±25399.999 mm ±999.9999 inch --	±9999.9999 mm ±421.99999 inch --
inch	0.5*10^{-5} [inch] [degr.]	±25399.99 mm ±999.999 inch --	±25399.999 mm ±999.9999 inch --	±25399.999 mm ±999.99999 inch --	±9999.9999 mm ±421.99999 inch --

Input system	Position control resolution	Input resolution 10^{-3} [inch] [degr.]	10^{-4} [inch] [degr.]	10^{-5} [inch] [degr.]	10^{-6} [inch] [degr.]
mm	0.5*10^{-1} [degr.]	-- -- ±99999.9 degr.	-- -- --	-- -- --	-- -- --
mm	0.5*10^{-2} [degr.]	-- -- ±999999.99 degr.	-- -- ±99999.99 degr.	-- -- --	-- -- --
mm	0.5*10^{-3} [inch] [degr.]	±253999.99 mm ±9999.999 inch ±99999.999 degr.	±253999.99 mm ±9999.999 inch ±99999.999 degr.	±107374.18 mm ±9999.999 inch --	-- -- --
mm	0.5*10^{-4} [inch] [degr.]	±253999.99 mm ±9999.999 inch ±9999.999 degr.	±253999.99 mm ±9999.9999 inch ±9999.9999 degr.	±107374.18 mm ±9999.999 inch ±9999.999 inch	-- -- --
mm	2*10^{-5} [inch]	±25399.99 mm ±999.999 inch --	±25399.99 mm ±999.9999 inch --	±25399.99 mm ±999.99999 inch --	±10737.418 mm ±999.99999 inch --
mm	0.5*10^{-5} [inch] [degr.]	±25399.99 mm ±999.999 inch ±999.999 degr.	±25399.999 mm ±999.9999 inch ±999.9999 degr.	±25399.999 mm ±999.99999 inch ±999.99999 degr.	±10737.418 mm ±999.99999 inch ±999.99999 degr.

± 99999.999 mm	Max. traversing range for linear axes in [mm]
± 3837.0078 inch	Max. traversing range for linear axes in [inch]
± 99999.999 degr.	Max. traversing range for rotary axes in [degrees] (with NC MD 572* bit 2=0)

Unit system	Position control resolution	Input resolution			
		10^{-2} [mm] [degr.]	10^{-3} [mm] [degr.]	10^{-4} [mm] [degr.]	10^{-5} [mm] [degr.]
mm	0.5*10^{-1} [degr.]	-- -- ±999999.9 degr.	-- -- ±999999.9 degr.	-- -- --	-- -- --
mm	0.5*10^{-2} [mm] [degr.]	±99999.99 mm ±3937.007 inch ±999999.99 degr.	±99999.99 mm ±3937.007 inch ±999999.99 degr.	±99999.99 mm ±3937.007 inch ±99999.99 degr.	-- -- --
mm	0.5*10^{-3} [mm] [degr.]	±99999.99 mm ±3937.007 inch ±99999.99 degr.	±99999.999 mm ±3937.0078 inch ±99999.999 degr.	±99999.999 mm ±3937.0078 inch ±99999.999 degr.	-- -- --
mm	2*10^{-4} [mm]	±9999.99 mm ±393.700 inch --	±9999.999 mm ±393.7007 inch --	±9999.9999 mm ±393.70078 inch --	±9999.9999 mm ±393.70078 inch --
mm	0.5*10^{-4} [mm] [degr.]	±9999.99 mm ±393.700 inch ±9999.99 degr.	±9999.999 mm ±393.7007 inch ±9999.999 degr.	±9999.9999 mm ±393.70078 inch ±9999.9999 degr.	±9999.9999 mm ±393.70078 inch ±9999.9999 degr.
mm	0.5*10^{-5} [degr.]	-- -- ±999.99 degr.	-- -- ±999.999 degr.	-- -- ±999.9999 degr.	-- -- ±999.99999 degr.

Unit system	Position control resolution	Input resolution			
		10^{-3} [inch] [degr.]	10^{-4} [inch] [degr.]	10^{-5} [inch] [degr.]	10^{-6} [inch] [degr.]
mm	0.5*10^{-1} [degr.]	-- -- --	-- -- --	-- -- --	-- -- --
mm	0.5*10^{-2} [mm] [degr.]	±99999.99 mm ±3937.007 inch --	±99999.99 mm ±3937.007 inch --	±99999.99 mm ±3937.007 inch --	-- -- --
mm	0.5*10^{-3} [mm] [degr.]	±99999.999 mm ±3937.0078 inch	±99999.999 mm ±3937.0078 inch	±99999.999 mm ±3937.0078 inch --	-- -- --
mm	2*10^{-4} [mm]	-- -- --	-- -- --	±9999.9999 mm ±393.70078 inch --	±9999.9999 mm ±393.70078 inch --
mm	0.5*10^{-4} [mm] [degr.]	-- -- --	-- -- --	±9999.9999 mm ±393.70078 inch --	±9999.9999 mm ±393.70078 inch --
mm	0.5*10^{-5} [degr.]	-- -- --	-- -- --	-- -- --	-- -- --

± 99999.999 mm	Max. traversing range for linear axes in [mm]
± 3937.0078 inch	Max. traversing range for linear axes in [inch]
± 99999.999 degr.	Max. traversing range for rotary axes in [degrees] (with NC MD 572* bit 2=0)

11.7.10 Influence on the display

Operator display

The axis position is displayed with the relevant, axis-specific number of decimal places. No distinction is made between linear and rotary axes when defining the number of decimal places.

The values for zero offset, working area limitation and scale centre are displayed in the input resolution. If this is not possible, decimal places are truncated in order to enable the display of the entire integer portion of the value.

Rotary axis values can be displayed as absolute or modulo values (depending on NC MD 560* bit 7).

Service display

The service display shows the absolute position in units of position control resolution. When viewing this display, it must be noted that the significance of the last decade depends on the axis-specific position control resolution.

With an endlessly turning rotary axis (NC MD 572* bit 2 = 0), when the 32 bit limit has been reached (1073741.824 degrees = 2982.61 resolutions with resolution 10^{-3} degrees) an offset calculation is performed so that the service display jumps from the maximum positive value to the maximum negative value plus the offset.

11.7.11 Influence on the modes/function

"Increment" mode

In ”increment” mode, the machine travels at the specified incremental speed . The feedrate is specified via machine data. The increment is derived from the mode (NC) 1, 10, 100, 1000, 10000) and from the display resolution of the relevant axis.

The table below shows the traversing path in mm, inches or degrees in dependence on the display resolution and the mode.

Increment \ Display resolution	Millimeters / Degrees / Inches				
	10^{-1} [degr.]	10^{-2} [mm] [degr.]	10^{-3} [mm] [degr.] [inch]	10^{-4} [mm] [degr.] [inch]	10^{-5} [degr.] [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

"DRF" function

In "DRF" mode, the handwheel pulses are also weighted with the display resolution of the selected axis. If the input resolution is coarsor than the display resolution (e.g. input resolution 10^{-2} mm, display resolution AF 10^{-3} mm) no DRF is possible.

"PRESET" function

The "Preset" mode can be used to shift control zero to an arbitrary point in the machine coordinate system. The Preset offset may comprise no more than 8 decades (plus sign). An alarm is issued if more than 8 decades are entered.
The following value ranges must be observed when specifying a Preset value for rotary axes.

Input resolution	Value range
10^{-2} degrees	0 ... 359.99
10^{-3} degrees	0 ... 359.999
10^{-4} degrees	0 ... 359.9999
10^{-5} degrees	0 ... 359.99999
10^{-6} degrees	0 ... 359.999999

"PLAY BACK" function

In this mode, position values approached in jog mode are taken over into the part program. The input resolution determines the number of decimal places with which the coordinate values are transferred to the part program. The number of decimal places is identical for all axes and is independent of the display or position control resolution. As many as 8 decades can be transferred as position value to the part program. Where applicable, decimal places are not transferred to the part program.

11.8 Data Protection

11.8.1 Data areas

The following data areas are battery-backed in the power supply unit:

- NC machine data
- Cycle machine data
- Cycle setting data
- PLC machine data
- Setting data
- Main programs
- Subroutines
- USM data (RAM)
- Tool offsets
- Zero offsets
- R parameters
- Compensation data

PLC data (RAM on PLC CPU):

- Data blocks
- User program (which is normally in EPROM)

The following may result in a loss of data:

- Battery failure or replacing of the battery with the control switched off
- Connection between battery in power supply unit and CPUs open
- NC CPU defective or disconnected (NC data)
- PLC CPU defective or disconnected (PLC data)
- COM CPU removed
- Multi-port/dual-port defective of disconnected

The data must be backed up as soon as all installation procedures have been completed (e.g. on floppy disk or punched tape). Particular care must be taken as regards the backup of machine data.

11.8.2 Data backup with the programmer

11.8.2.1 Introduction

Using the PG 675, PG 685 or PG 750 and the appropriate TRANS-PGIN/PCIN software, all data and programs of the SINUMERIK 880 GA2 can be input and output via the V.24 (RS 232 C) interface. Part programs can be developed and documented using the ED, WordStar or VEDIT editor.

This Section describes the operating sequence on the SINUMERIK 880 GA2, data handling using the PG 675, PG 685 or PG 750 with the appropriate operating system and TRANS-PGIN/PCIN.

The assignment of operating systems to programmers and the order numbers of the appropriate TRANS-PGIN/PCIN software are shown in the following table:

Program-mer	Operating system	Interface	Order No. of German Trans-PGIN/PCIN	Order No. of English Trans-PGIN/PCIN
PG 635	PCP/M	Printer	6FC3 981 - 7AM	6FC3 981 - 7BM
PG 675	CP/M	Printer	6FC3 981 - 7AJ	6FC3 981 - 7BJ
PG 685	PCP/M-86	Printer	6FC3 981 - 7AL	6FC3 981 - 7BL
PC 16-11	PCP/M-86	Printer	6FC3 981 - 7AK	6FC3 981 - 7BK
PG 685	MS-DOS 2.11	Printer	6FC3 981 - 7AP	6FC3 981 - 7BP
PC 16-20	MS-DOS 3.1	COM 1	6FC3 981 - 7AN	6FC3 981 - 7BN
PG 750	MS-DOS 3.2	COM 2	6FC3 981 - 7AN	6FC3 981 - 7BN
PC 32-05	MS-DOS 3.2	COM 3	6FC3 981 - 7AR	6FC3 981 - 7BR
PCIN version 2.0 with operator guidance				
PG 730	PCP/M	AUX0 / AUX1	6FC3 981 - 7CA 00*	
PG 750	PCP/M	AUX0 / AUX1	6FC3 981 - 7CA 00*	
PG 730	MS-DOS	COM1 - COM4**	6FC3 981 - 7CB 00*	
PG 750	MS-DOS	COM1 - COM4**	6FC3 981 - 7CB 00*	
PCs	MS-DOS	COM1 - COM4**	6FC3 981 - 7CB 00*	

* In several languages (English, German, French, Italian, Spanish)

** The number of COMs depends on the configuration of the programmer

11.8.2.2 Using diskettes

The operating system is loaded immediately after the programmer is switched on.

PG 675: CP/M-86: The operating system is loaded from the system diskette inserted in drive A:.

PG 685: PCP/M-86:
PG 750: MS-DOS 3.10: The operating system is loaded from the hard disk. However, if there is a diskette in drive A:, the programmer will try to load the operating system from it.

Formatting empty disks

A diskette must be formatted before data can be stored on it (Note: the write protection must not be in position either).

PG 675: CP/M-86

Insert the diskette to be formatted into drive B: and format it with the program FORMAT.

Call:	A> **FORMAT**	**Return** key

Response: CP**/**M86 DISK FORMAT UTILITY VERSION 1.0
Type "C" to cancel
Unformatted disk in drive B: ? (Y**/**N)

Input: **Y** or **N** (Yes**/**No)
When Y has been entered the system starts the formatting routine for the disk in drive B:

Response: Format started

After the routine has finished, a prompt asks whether you wish to format another diskette. Enter N for "no" and press return key to leave the program.

PG 685: PCP/M-86

Insert the diskette to be formatted into drive A: and format it with the program DSKMAINT.

Call :	B> **DSKMAINT**	**Return** key

The main menu of this program appears on the screen. Insert the diskette to be formatted into drive A: and then press **f7** to select formatting. The formatting menu appears on the screen. Press **f1** to select the diskette in drive A: for formatting.

Response: Formatting disk in drive A:
This operation will erase all data on drive A:
Is this what you want? (Yes **/**No)

Input: **Y** or **N** (Yes**/**No)
When Y has been entered the system starts the formatting routine for the disk in drive A:

After the routine has finished, a prompt asks whether you wish to format another diskette. Enter N for "no" and press return key to leave the formatting function and return to the main menu.

PG 750: MS-DOS 3.10

Format the diskette with the program **FORMAT**.

Call:

– for 1.2 MB:	C> **Format A: /V**	**Return** key
– for 360 KB:	C> **Format A: /4 /V**	**Return** key

Response: Insert new diskette for drive A: and strike RETURN when ready.
When you have pressed the RETURN key, the system starts the formatting routine for the diskette.

Response: Formatting ...-
When formatting is complete the following message appears:
Formatting ... Format complete

Prompt: Volume label
(11 characters, ENTER for none)

After you have entered the volume label and/or pressed the RETURN key, a prompt asks whether you wish to format another diskette. Enter N for "no" and press the RETURN key to leave the formatting function.

Copying diskettes

For safety reasons it is always advisable to make two diskettes (one working diskette and one backup copy).

PG 675: CP/M-86

You can copy diskettes with the program **COPYDISK**.

Call: A> **COPYDISK** **Return** key

Response: CP/M86 FULL DISK COPY Utility
Version 2.0
Enter Source Disk (A-D) ?

Input: A(B) (source drive)

Response: Destination Disk Drive (A-D) ?

Input: B(A) (destination drive)

Response: Copying Disk A(B): to Disk B(A):
Is this what you want to do (Y/N)?

Input: **Y** or **N** (Yes/No)
When Y has been entered the system starts the copying routine
and A(B) is copied to B(A).

After the routine has finished a prompt asks whether you wish to copy another diskette. Enter N for "no" and press the RETURN key to leave the copying function.

PG 685: PCP/M-86

You can copy diskettes with the program **DSKMAINT**.

Call: B> **DSKMAINT** **Return** key

The main menu of this program appears on the screen. Press **f3** to select copying. Press **f1** twice to select both the source and the destination drives (there is only one drive).

Response: Copying from diskette in drive A:
to diskette in drive A:
This operation will erase all data on drive A:
Is this what you want? (Yes/No)

Input: **Y** or **N** (Yes/No)

Response: Copying from diskette in drive A:
to diskette in drive A:
INSERT SOURCE DISKETTE AND PRESS ANY KEY

Insert the diskette to be copied and press any key.

Response: Copying from diskette in drive A:
to diskette in drive A:
INSERT DESTINATION DISKETTE AND PRESS ANY KEY

Insert an empty diskette and press any key.

The diskette is copied in several parts. The diskette to be copied (source) and the empty diskette (target) must therefore be inserted one after the other several times. When copying is complete a prompt asks whether you wish to copy another diskette. Press **f8** to leave the copying function and return to the main menu.

PG 750: MS-DOS 3.10

You can copy diskettes with the program **DISKCOPY**.

Call: C> **DISKCOPY A: B:** **Return** key

Response: Insert SOURCE diskette into drive A:

Insert TARGET diskette into drive B:

Press any key when ready.

Insert the diskette to be copied into the source drive A: and a formatted empty diskette into the target drive B:. When you press any key the program starts copying.

After the routine has finished, a prompt asks whether you wish to copy another diskette. Enter N for "no" and press return key to leave the copying function.

Note:

If you only have one drive it is both the source and the target drive while copying. In this case, the diskette is copied in several parts. The diskette to be copied (source) and the empty diskette (target) must therefore be inserted one after the other several times.

Changing diskettes

In the PG 675, every time a diskette is changed, the newly inserted diskette must be announced with CTRL C if you are going to write to it. If you do not do this, edited files cannot be stored because changing diskettes activates a write protection (message: Bdos Error R/O).

In the PG 685 and PG 750 you do not need to announce a newly inserted diskette with CTRL C after a diskette change in order to be able to write to it.

Listing the directory

With the following functions the hard disk or diskette directory can be listed on the screen or printed out (hardcopy key) and the diskette status can be checked.

PG 675: CP/ M-86

Call:		A> **DIR**	**Return** key
	or	A> **DIR B:**	**Return** key

The files on the diskette in drive A: or B: are listed without any indication of their size.

Call:		A> **STAT *.***	**Return** key
	or	A> **STAT B:*.***	**Return** key

The files are listed with an indication of the size of each as well as the total memory occupied and the free space on the diskette. The diskette status "RW" (READ**/**WRITE) or "RO" (READ ONLY) is also given. An empty diskette contains 340 Kbytes.

PG 685: PCP/M-86

Call:		B> **DIR**	**Return** key
	or	B> **DIRS**	**Return** key
	or	B> **DIR A:**	**Return** key
or	B> **DIRS A:**		**Return** key

The files stored on the hard disk or on the diskette in drive A: are listed without any indication of their size. With the DIR command the names of files with the DIR attribute are listed. Files with the SYS attribute are listed with the DIRS command.

Call:		B> **DIR [SIZE]**	**Return** key
	or	B> **DIR A: [SIZE]**	**Return** key

The files are listed with an indication of the size of each as well as the total memory occupied on the hard disk or the diskette in drive A:.

Call:		B> **SHOW**	**Return** key
	or	B> **SHOW A:**	**Return** key

Response: e.g. A: RW, free space 84K

You can display the free space on the hard disk or diskette with the command SHOW. The status "RW" (READ**/**WRITE) or "RO" (READ ONLY) is also shown. An empty diskette contains 694 Kbytes.

PG 750: **MS-DOS 3.10**

Call: B(C)> **DIR [/p] [/w]** **Return** key

or B(C)> **DIR A(B): [/p] [/w]** **Return** key

The files stored in the current directory of the hard disk or on the diskette in drive A(B) are listed. For every file the size in bytes, the date and the time of last processing are specified. The free space and the number of files in the directory are also displayed.

An empty diskette for the PG 750 contains 1.2 MB or 360 KB.

The DIR command has two options:

The **/**p option activates page mode. Output of the directory is delayed after each screen page until any key is pressed.

The **/**w option activates the wide display in which the file designations are output without additional information. Five files are displayed per line.

11.8.2.3 File handling

Files are stored on diskette or hard disk. They are addressed via their file designations.

The full file designation consists of three parts:

1. Name of the drive
2. File name
3. Extension

Example: B> **VEDIT A:TESTPROG.SY8**

Extension

File name

Name of the drive

The file name consists of 1 to 8 characters. The file name is separated from the extension by a full stop. The extension consists of 1 to 3 characters. The file name and the extension can be freely selected. They can contain any characters except certain special characters (see the Quick Reference publication for to the operating system of your programmer).

The drive name is always placed in front of the file name when the file addressed is not on the current drive. If the file is on the current drive the name of the drive is not required.

Listing files

The command file TYPE is used to display the contents of any file on the screen. The command makes it easier to verify the contents of a file because you do not need to run an editor.

TYPE <drive>:<file designation> **Return** key

Example: **TYPE A:PC.DOK**

If the file is large you can stop the screen and restart it.

PG 675:	CP/M-86	stop with CTRL S, restart with CTRL Q
PG 685:	PCP/M-86	stop with CTRL S, restart with CTRL W

On the PG 750 you can display the file page by page if you use the following call:

TYPE <drive> : <file designation> **|** More **Return** key

Example: TYPE A: PC.DOK **|** More

Copying files

You can copy files with the command file PIP or Copy.

- Copy keeping the old name
  PG 675: CP/M-86

A>PIP B:=A:TEST.880	copy from A: to B:
A>PIP A:=B:TEST.880	copy from B: to A:

  PG 685: PCP/M-86

B>PIP B:=A:TEST.880	copy from A: to hard disk
B>PIP A:=B:TEST.880	copy from hard disk to A:

  PG 750: MS-DOS 3.10

C>COPY TEST.880 A:	copy from hard disk C: to drive A:

- Copy and rename
  PG 675: CP/M-86

A>PIP TEXT.NEU=TEXT.ALT	on diskette in drive A:
A>PIP B: TEST.1=B:TEST0	on diskette in drive B:

  PG 685: PCP/M-86

B>PIP TEXT.NEU=TEXT.ALT	on hard disk
B>PIP A:TEST.1=A:TEST0	on diskette in drive A:

  PG 750: MS-DOS 3.10

C>COPY TEXT.ALT = TEXT.NEU	on hard disk
C>COPY A:ALT.TXT = A:NEU.TXT	on diskette in drive A:

- Peripheral device as the destination (NC or printer)
  PG 675: CP/M-86

A>PIP LST:=B:TESTPROG.880	file on diskette in drive B:

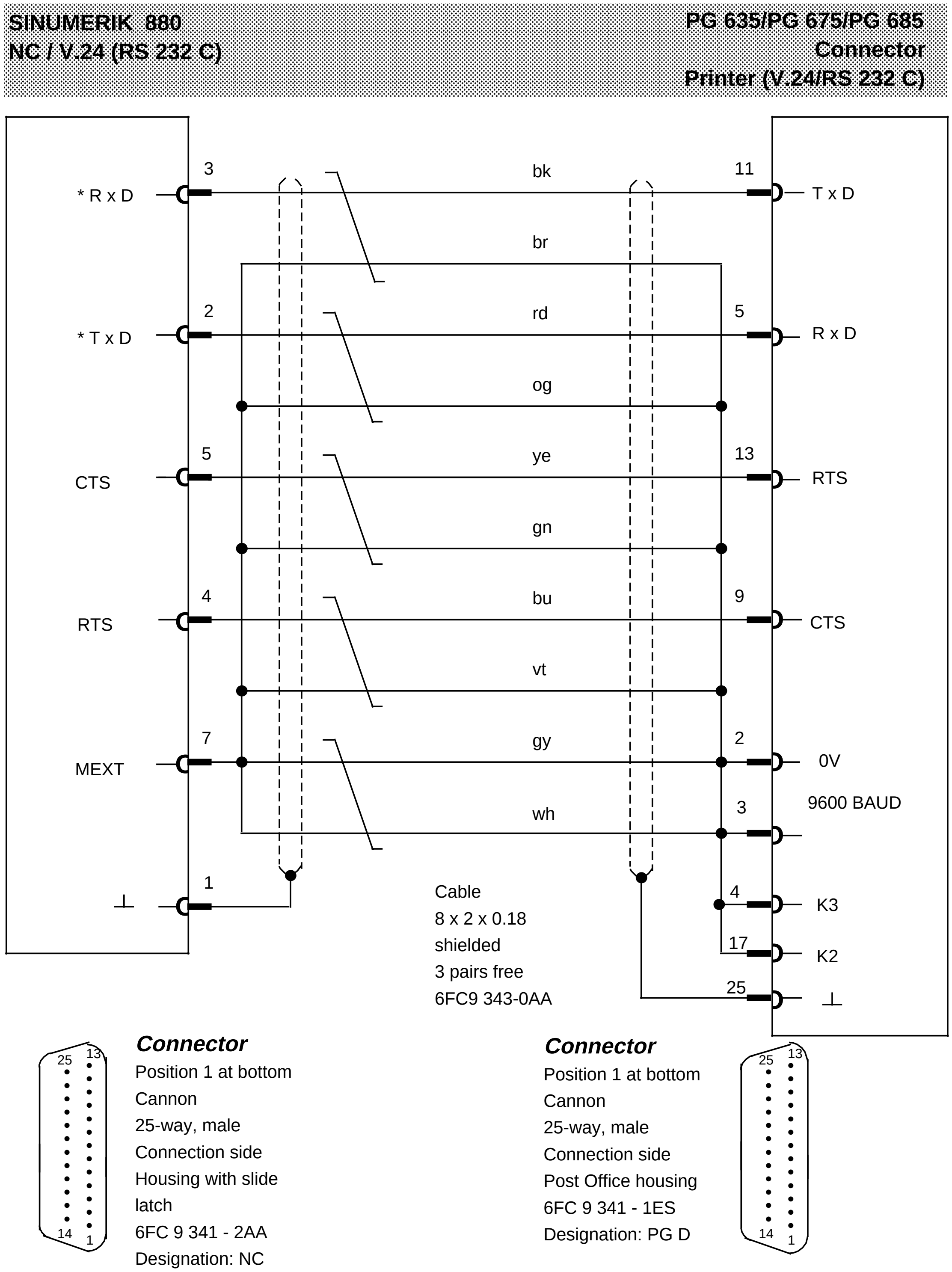
  PG 685: PCP/M-86

B>PIP LST:=A:TESTPROG.880	file on diskette in drive A:

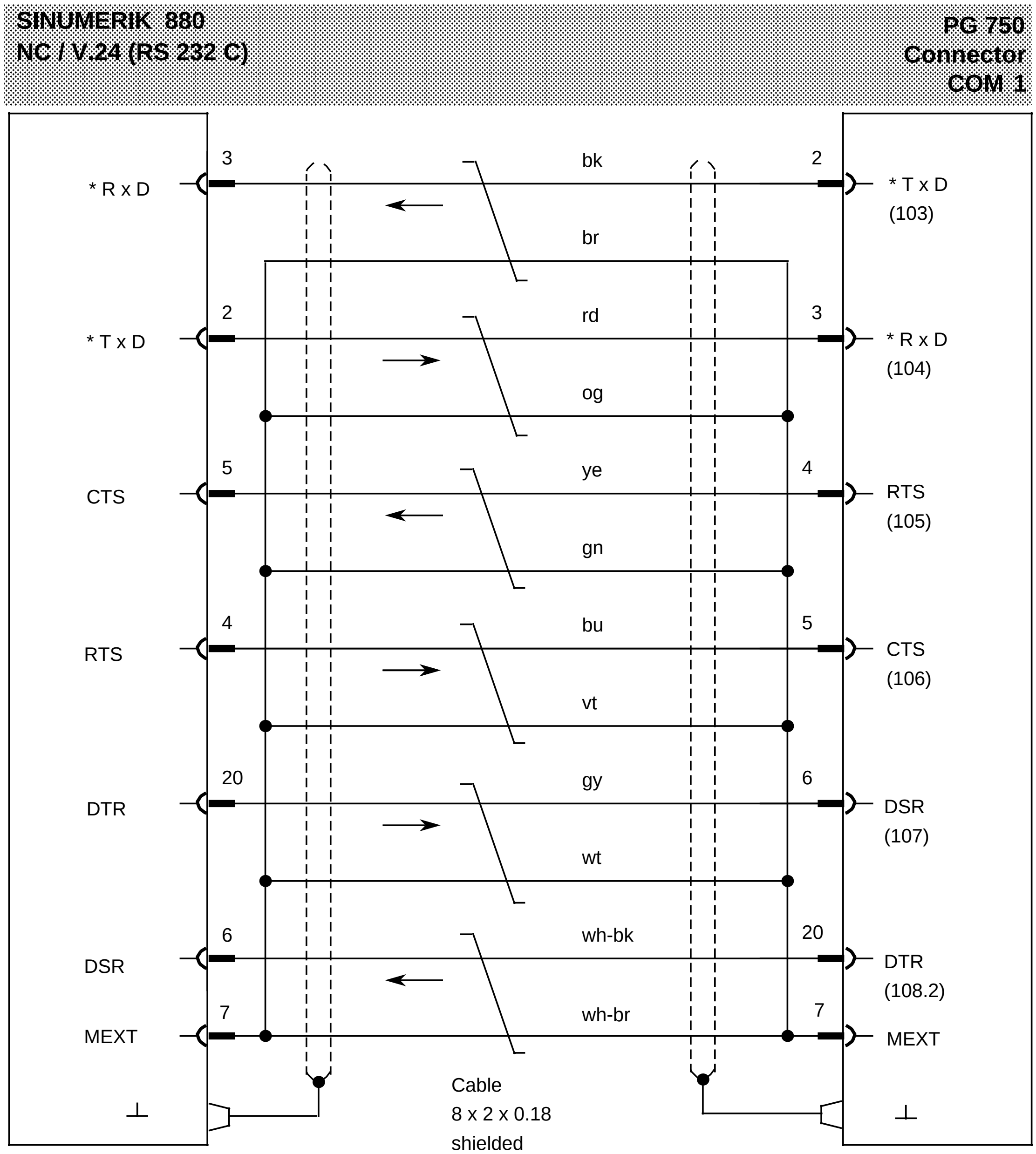
  PG 750: MS-DOS 3.10

C>COPY TESTPROG COM1	file on hard disk
C>COPY A:TESTPROG COM1	file on diskette on drive A:

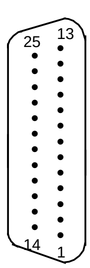
Cable name : SIMATIC PG 635/PG 675/PG 685 (TRANS-PG IN)
Order No. : 6FC9 344-1A



Cable name : SIMATIC PG 750 (TRANS PC IN and PLC programming)
PC (AT compatible)
Order No. : 6FC9 344-4R



Cable
8 x 2 x 0.18
shielded
2 pairs free
6FC9 343-0AA



Connector

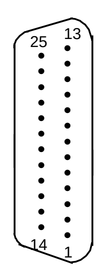
Position 1 at bottom
Cannon
25-way, male
Connection side
Housing with slide latch
6FC9 341-2AA
Designation: NC

Connector code

● Coding pin
× Without coding pin

Connector

Position 1 at bottom
Cannon
25-way, male
Connection side
Post Office housing
6FC9 341-1ES
Designation: PC



Connector code

● Coding pin
× Without coding pin

11.8.2.4 Interface settings

Interface parameters on the SINUMERIK 805, 810, 820, 840, 880

How to input and modify the interface parameters is described in the Operator's Guide of your CNC control.

Interface parameters on the programmer

On the PG 675 the printer interface does not need to be initialized.

Before the printer interface is called on the PG 685, data transfer to the NC must be initialized to 7 data bits and even parity in the following manner:

DEVICE LPT0 [DAT=7]	(7 data bits)	Return key
DEVICE LPT0 [PAR=EVEN]	(even parity)	Return key

These inputs are not necessary if the **STARTUP.SUB** file supplied is copied onto the system diskette of the PG 635 or the hard disk of the PG 685. The two DEVICE commands are automatically executed by the **STARTUP.SUB** file whenever the operating system is loaded.

On the PG 750 the interface is initialized by calling the **MODE** command.

Examples: MODE COM1: 9600, e, 7, 2, p (PG 750)

2 – Stop bits
7 – Data bits
e – Parity
9600 – Baud rate

This call is not required if the command is written in the AUTOEXEC.BAT file. The interface is then initialized whenever the operating system is restarted.

Other baud rates (for PG 635, PG 675, PG 685 only)

When using the data cable as shown in 10.4.1 the baudrate is set to 9600 baud as standard. For other baud rates the connector to the printer interface must be changed as follows:

	Pins (X pin connected to pin 2)		
Baud rate	3	4	17
9600	X	X	X
4800	X	0	X
2400	0	X	X
1200	X	0	0
600	0	X	0
300	X	X	0
110	0	0	X

Data start identifiers

When the NC outputs data it transmits initial identifiers to indicate the type of data. When reading in from the PG 675 or PG 685 **/** PG 750 these initial identifiers must be available in the file in the same form.

% MPFxxxx	CR LF	Identifier for main programs
% SPFxxx	CR LF	Identifier for subroutines
% T E A 1	CR LF	Identifier for NC machine data
% Z O A	CR LF	Identifier for zero offsets
% T E A 2	CR LF	Identifier for PLC machine data
% R P A	CR LF	Identifier for R parameters
% S E A 1-4	CR LF	Identifier for setting data
% PCF	CR LF	Identifier for PLC alarm texts
% IKA 1-3	CR LF	Identifier for compensation data

11.8.2.5 How to transfer from the SINUMERIK 880 GA2 to the programmer

a) Cable connection to PG 675**/**PG 685: V.24 (RS 232) interface PRINTER
PG 750: V.24 (RS 232) interface COM1

b) Cable connection to NC: V.24 (RS232) interface (1 to 4)

c) Load operating system:

CP**/**M-86	PG 675
PCP**/**M-86	PG 685
MS-DOS	PG 750

d) Initialise printer interface

e) Select DATA TRANSMISSION dialog and set the following parameters:

Device type: RTS-LINE
Baud rate: 9600
Parity: none
Stopbits: 2

Branch to the DATA OUTPUT dialog with the softkey Read Out.

f) Select the data type to be transferred as described in the Operator's Guide.

g) Program call for transfer:

PG 675:A>PGIN <drive>:<file designation>
PG 685:B>PGIN <drive>:<file designation>
PG 750:C>PCIN <drive>:<file designation>

h) Start the PG by pressing the RETURN key

i) Start the SINUMERIK 880 with softkey: Start output

Now the specified programs of the NC are stored on the specified drive in the programmer under the specified file designations.

During transmission the character received is output on the PG screen and checked for overflow and step errors (e.g. due to incorrect baud rate). If such an error occurs transmission is aborted and an error text displayed. The PGIN and PCIN data receive programs can store programs with a length of up to 256 Kbytes.

The reception of data is terminated if:
1. 40 times 00hex was transmitted (output from NC)
2. The "***" key was pressed on the PG 675 or 685 or the "ESC" key was pressed on the PG 750.
3. The number of received characters exceeds the 256 Kbyte limit.

If the first character (after 00hex) is a % or CR character the data start identifier is evaluated and the data type shown on the screen of the programmer after transmission.

After transmission of main programs or subroutines a directory of the transmitted program numbers and the length of each program is created automatically (see examples). The characters are counted in the same way as in the SINUMERIK.

After transmission of other data the data type is displayed but no directory created.

If the data start identifier does not correspond to a known string the following message is displayed after transmission:
"unknown data"

However, the characters received are still stored on hard disk or diskette.

Examples:

Program call	Storage		
	In drive	File name	Directory name
PG 675			
A>PGIN NAME	A:	NAME	NAME.DIR
A>PGIN B:NAME	B:	NAME	NAME.DIR
B>PGIN NAME	B:	NAME	NAME.DIR
B>PGIN A:NAME	A:	NAME	NAME.DIR
PG 685			
A>PGIN NAME	A:	NAME	NAME.DIR
A>PGIN B:NAME	B:	NAME	NAME.DIR
B>PGIN NAME	B:	NAME	NAME.DIR
B>PGIN A:NAME	A:	NAME	NAME.DIR
PG 750			
C>PCIN NAME	C:	NAME	NAME.DIR
C>PCIN A:NAME	A:	NAME	NAME.DIR
C>PCIN B:NAME	B:	NAME	NAME.DIR

If a file name is specified with an extension the extension DIR is not permissible because the directory is stored under that designation.

With the call type <drive>:<file designation>.DIR you can display the directory of the main programs and subroutines transmitted.

Example:

B>**TYPE A:TEST-L.DIR**

L	95	1609	CH	L	97	1178	CH	L	98	310	CH	L	801	33	CH
L	803	42	CH	L	804	42	CH	L	805	58	CH	L	806	90	CH
L	951	75	CH	L	970	107	CH	L	971	104	CH	L	981	58	CH
L	990	54	CH	L	999	12	CH								

Example:

B>**TYPE A:TEST-%.DIR**

% 120 105 CH % 22 105 CH

11.8.2.6 How to transfer from the programmer to the SINUMERIK 880 GA2

a) Cable connection to PG 675/PG 685: V.24 (RS 232) interface PRINTER
PG 750: V.24 (RS 232) interface COM1

b) Cable connection NC: V.24 (RS 232) interface (1 to 4)

c) Load operating system:

CP/M-86	PG 675
PCP/M-86	PG 685
MS-DOS	PG 750

d) Initialise printer interface

e) Select DATA TRANSMISSION dialog and set parameters:

Device type:	RTS-LINE
Baud rate:	9600
Parity:	none
Stop bits:	2

Branch to the DATA OUTPUT dialog with the softkey Read Out.

f) Program call for transfer:

PG 675:A>PIP LST:=<drive>:<file designation>[E]
PG 685:B>PIP LST:=<drive>:<file designation>[E]
PG 750:C>COPY<drive>:<file designation>COM1

[E]: with display of the characters on the screen

g) Start the SINUMERIK 880 with softkey: Start input

h) Start the PG by pressing the RETURN key.

Now the specified file is loaded into the NC.

If several files must be transferred to the NC, the PIP program can be loaded to avoid repeated PIP system calls on the PG 675 and PG 685.

Example:

1st part	B>**PIP**	**RETURN** key
2nd part	***LST:=<DRIVE>:<FILE DESIGNATION>**	**RETURN** key

Only the second part of the call need be repeated with a new file name.

After termination of transmission you can leave the "PIP SYSTEM" by pressing the RETURN key.

11.9 Remote Programmer Function

Activation of the remote programmer function:

- Requirements
  - Hardware

    SINEC H1 industrial LAN
    PG 750 with CP 141 or PG 685 with CP 536
    CP 231 A
    135 WB with ACOP

  - Firmware

    COM software version 1.1
    PLC software version 1.1
    CP firmware version 3A (Order number: 6FX1841-0BX02-3A)

  - Machine data

    On the SINUMERIK 880 GA2 the function remote programmer via SINEC H1 is only disabled in the option bit. This NC machine data bit must be set in order to establish a link between the remote programmer and the SINUMERIK 880 GA2.

    The remote programmer function of the SINUMERIK is adapted to the remote programmer function of the SIMATIC. However, the following points must be observed.

    If a user is to establish a link the valid Ethernet address of the CP 231 must be known to him.

    If both the remote programmer functionality and the computer link are active, the Ethernet address can be seen from the data base produced with the NML configuration tool. The Ethernet address is stored in the "node and interface description" menu.

    If a computer link functionality is not used the Ethernet address is stored in machine data bits 5141 to 5146.

    The password is stored in the general NC machine data 234 and 235. If a zero is stored in NC machine data 234 and 235, a password is not required (press CR at the password prompt). If NC machine data 234 contains the value 41424344 and NC machine data 235 the value 45464748, the password ABCDEFGH must be entered. Input into NC machine data 234 and 235 is protected by the password of the SINUMERIK 880 GA2. The value entered in these machine data is interpreted as ASCII.

  - Establishing a link between the remote programmer and the SINUMERIK CP

    Start the S5 DOS at the programmer.
    Select the SINEC H1 interface with the "interface" menu.
    Select "utility" menu
    Select "bus selection" menu.
    Select "edit path" menu.

    Here the path CP 536 - CP 5365 - Corr/Mux - ENDP must be edited.

    Now the valid Ethernet address of the CP 231 A must be input into the Ethernet address line. Then the password is requested.

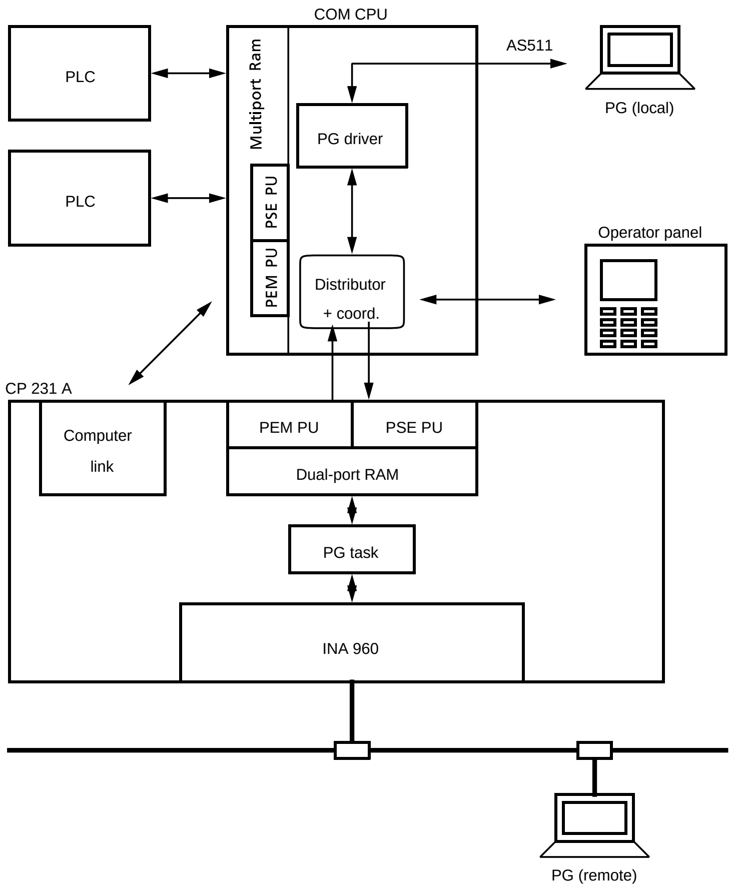
    At "address" the number of the PLC with which the remote programmer is to communicate is requested.

    Now transfer the data.

Select "active" menu.

The function key "all" is now pressed to establish the link. The message "link established" appears.

Now return to the basic menu and select "package". From now on operation is the same as with the local programmer.



2. Abort of the remote programmer function by activation of the local programmer.

   To activate the local programmer, press the following softkeys (the shaded softkeys are the ones to press).

   Enter "00000100" into the setting data 5030 so that the 3rd interface is set to programmer mode.

   AUTOMATIC mode

TOOL OFFSET	SETTING DATA	DATA IN-OUT	PART PROGRAM	ACTUAL BLOCK	>

   Enter a "3" in the input screen for the third interface.

INTERF. 1+2	INTERF. 3+4	FILE-TRANSF.		

INPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

   Then the prompt "**REMOTE PG RUNNING/STOP?**" appears.

   Now the user must decide whether he wishes to break off the remote programmer connection. If he does not, he can leave this screen and the remote programmer connection will have been left intact.

   If the local programmer is to override the remote programmer, i.e. the remote programmer breaks off its connection to the PLC, press the following key:

INPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

   Now softkey function 143 is executed. If a remote programmer is active, the driver of this function on the COM CPU is made to break off the link between the remote PG and the PLC in the SINUMERIK via the CP231 and CPM CPU components. If this is successful, the message "**REMOTE PG IS INACTIVE**" is displayed.

   If the link is not broken within a set time the message "**CHECK REMOTE PG**" is output. In this case you should check the COM CPU and the CP231 and reinitialize them if necessary.

   When the message "**REMOTE PG IS INACTIVE**" appears you can press the softkey again and the local programmer is deactivated. If you press the softkey yet again, the usual message "interface busy" appears.

iNPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

You have to enter 00000100 into SD 5030 to set the third interface to programmer mode.

To stop the remote programmer follow the following procedure (pressing the shaded softkeys).

Automatic mode

TOOL OFFSET	SETTING DATA	DATA IN-OUT	PART PROGRAM	ACTUAL BLOCK	>

Enter a "3" in the input screen for the third interface.

INTERF. 1+2	INTERF. 3+4	FILE-TRANSF.		

Now press the following softkey:

INPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

Then the prompt "**REMOTE PG RUNNING/STOP?**" appears, if a remote programmer is active.

Or the message "**REMOTE PG IS INACTIVE?**" if no remote programmer is running. In this case you can press the softkey INPUT START immediately to activate the third interface for local programmer mode.

If the prompt "**REMOTE PG RUNNING/STOP?**" appears the user must decide whether he wishes to break off the remote programmer connection. If he does not, he can leave this screen and the remote programmer connection will have been left intact.

If you do not intend the local programmer to override the remote programmer, i.e. the remote programmer is not supposed to break off the link to the PLC, you have to press the following softkey again.

INPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

Then proceed as described above.

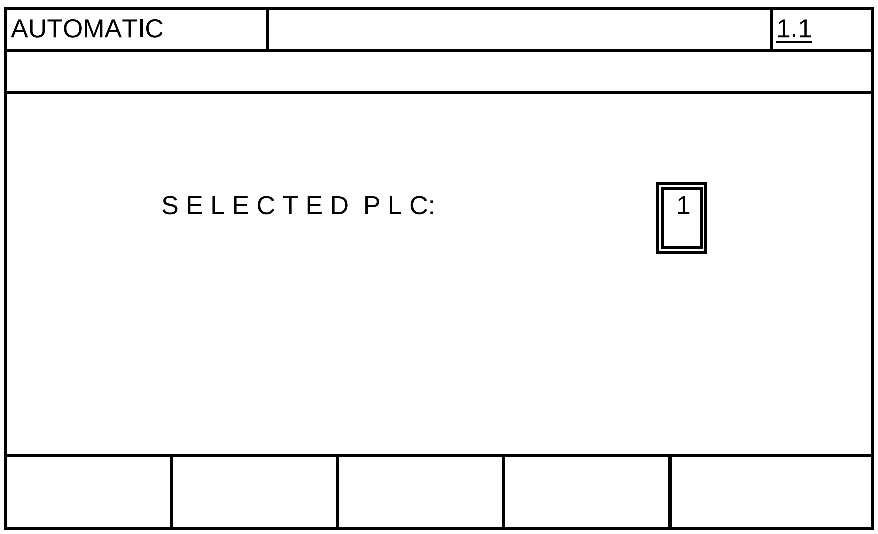
3. Input modifications in the "selected PLC" screen because of the remote programmer function.

   There is a display for the PLC number which you can reach via the following softkeys:

BLOCK SEARCH	PROGRAM CONTROL	DIAG- NOSTICS	OVER- STORE:		>

ALARM MESSAGE	MACHINE DATA	SERVICE STATUS	PLC UTILTIY		>

   The screen looks like this:



   You can enter the PLC numbers into the input field shown here in the double box. Up to two PLCs are possible. The permissible inputs are therefore 1 and 2. If a larger number is entered the message "GENERAL DATA ERROR" is displayed. The message "GENERAL DATA ERROR" also appears if you try to select the second PLC although a second PLC is not declared in NC machine data 5038.

   The response of the control to input in this screen changes only when a remote programmer is active. Then it is no longer possible to change the PLC number on this screen. If you try to enter the number in this screen the input is not accepted and the message "**REMOTE PG RUNNING/STOP?**" is displayed. The input is refused because otherwise the remote programmer would be connected with another PLC that was not specified by the operator of the remote programmer but by the operator of the control. If the remote programmer breaks off its link the PLC number which was entered in this screen before activation of the remote programmer is entered.

   If the local programmer has been activated you can enter the PLC number as described previously.

4. Operation in initial clear mode or installation

Here new or modified operating sequences of the 880 control are described which have arisen because of the remote programmer function.

The remote programmer function can be activated during installation as well.

For this reason the same problems arise as in normal mode. Before starting the local programmer the remote programmer must first be deactivated. This is done in the same way as in normal mode. The only difference is in the softkey menus. The softkey menus of normal mode are shown with their equivalents in initial clear mode. Operation is as described above.

Now press the following softkey:

Initial clear mode

DATA IN-OUT	NC DATA	PLC UTILITY	MACHINE DATA	SET UP END PW	>

Normal mode in AUTOMATIC mode

TOOL OFFSET	SETTING DATA	DATA IN-OUT	PART PROGRAM	ACTUAL BLOCK	>

Initial clear mode

INTERF. 1+2	INTERF. 3+4			

Normal mode in AUTOMATIC mode

INTERF. 1+2	INTERF. 3+4	FILE-TRANSF.		

Initial clear mode

DATA IN	DATA OUT			STOP ALL

Initial clear mode

		REMOTE PG STOP	START	STOP

Normal mode in AUTOMATIC mode

INPUT START	DATA OUTPUT	REMOTE PG STOP	STOP	STOP ALL

The only real difference between operation in normal mode and in initial clear mode is that you have to press the softkeys DATA INPUT and START in initial clear mode while just pressing the softkey INPUT START in normal mode has the same effect.

Changes were made in the function PLC initial clear as well.

This function is activated via the following softkeys.

Installation mode

DATA IN-OUT	NC DATA	PLC FUNCT.	MACHINE DATA	SET UP END PW	>

PLC INITIAL				

PLC 1 INITIAL	PLC 2 INITIAL			

After this operating sequence the function PLC initial clear is executed and a checkmark appears to indicate that the function is running correctly. This is how it worked until now and it still does as long as a programmer (remote or local) is not active. For a local programmer active means that the third interface is set to programmer mode and has been activated by pressing the INPUT START softkey. The remote programmer is activated when the operator of the remote programmer sends an activation message frame to the SINUMERIK 880 GA2 and receives a positive acknowledgement.

If a programmer is active the message "**STOP PG**" is displayed. The cancel alarm 3151 "Interface occupied by PG" is also output. It is to be seen in the NC alarm display. To obtain this display from the PLC 1/2 initial clear display, press the following keys:

^

^

>

NC ALARM				>

The screen looks like this:

		1.1
2000	ORD1	EMERGENCY stop
NC ALARMS		
2000	ORD1	EMERGENCY stop
3151	ORD2	Interface occupied by PG

NC ALARM				>

5. Error message

If the COM CPU detects an error in the remote programmer on the CP231 the cancel alarm **3165 "Remote PG has failed"** is issued at the operator panel. This alarm is output cyclially at approx. 5-second intervals. The cause of this alarm is in the SINEC module. To remedy the fault you have to check the CP231. Check the software version of the CP231 and if necessary clear the dual-port RAM to the CP231. This is done in installation mode where the dual-port RAM to the CP231 is cleared automatically.

11.10 Coordinate transformation

The coordinate transformation TRANSMIT is used in the face milling of turned parts (lathes). In order to implement this, a C axis and a powered milling cutter are required in addition to the X and Y axes.

The 2D/3D coordinate transformation is required when surfaces are configured on a plane (2D coordinate transformation) or in space (3D coordinate transformation) in such a way that they can only be processed by the (real) axes by including rotations (the tool must generally be located on the processing plane).

11.10.1 Corresponding data

- NC MD 730 1st, transformation, parameter 1
  NC MD 731 1st, transformation, parameter 2
  NC MD 732 1st, transformation, parameter 3
  NC MD 733 1st, transformation, parameter 4
  NC MD 734 1st, transformation, parameter 5
  NC MD 735 1st, transformation, parameter 6
  NC MD 736 1st, transformation, parameter 7
  NC MD 737 1st, transformation, parameter 8
  NC MD 738 1st, transformation, parameter 9
  NC MD 739 1st, transformation, parameter 10
- NC MD 740 to 809 2nd, to 8th, transformation, parameter 1 to 10
- NC MD 5060 channel numbers of the transformation
  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 3rd, fictitious axis
  NC MD 5065 axis name 1st, real axis
  NC MD 5066 axis name 2nd, real axis
  NC MD 5067 axis name 3rd, real axis
  NC MD 5068 axis name 4th, real axis
  NC MD 5069 axis name 5th, real axis
  NC MD 5070 to 5139 1st, to 8th, transformation data set
- MD 540* Bit 7 no transformation cancellation
- MD 564* Bit 6 fictitious axis
- OPTION transformation TRANSMIT/2D/3D coordinate transformation
- SIGNAL DB10-DB13 DR13 BIT 7 TRANSFORMATION ACTIVE
- Alarm 2043 Programming error during transformation
- Alarm 2189 Transformation undefined
- Alarm 2190 Transformation axes assigned
- Alarm 3086 Illegal transformation selection
- Alarm 3087 Error in transformation data

11.10.2 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. Up to eight transformations with varying axis groupings can be defined in any one control. The definition consists of a transformation data set and the associated parameters for each transformation.

The selection of the coordinate transformation is realized via G functions in the part program or via the command channel from the PLC.

The following G functions are specified:

G131, G231, G331 coordinate transformation TRANSMIT
G133, G233, G333 2D coordinate transformation
G135, G235, G335 3D coordinate transformation

Coordinate transformation can be cancelled via G130, G230 or G330 in the part program, or via the command channel from the PLC.

It can be specified via NC MD 540* Bit 7 whether cancellation occurs automatically on RESET or after a change in the operating mode.

See Section 11,8,8,1 for applications of the coordination transformation TRANSMIT.
2D coordinate transformation - see Section 11.8.8.2
3D coordinate transformation - see Section 11.8.8.3
and in the Programming Guide to System 880.

11.10.3 The transformation data set

NC MD	Bit No. 7	6	5	4	3	2	1	0
5060		Channel number of transformation						
5061		G function for transformation selection						
5062		Axis name 1st fictitious axis						
5063		Axis name 2nd fictitious axis						
5064		Axis name 3rd fictitious axis						
5065		Axis name 1st real axis						
5066		Axis name 2nd real axis						
5067		Axis name 3rd real axis						
5068		Axis name 4th real axis						
5069		Axis name 5th real axis						

NC MD 5060 to 5069 1st transformation data set
NC MD 5070 to 5079 2nd transformation data set
,
,
NC MD 5130 to 5139 8th transformation data set

Conditions for a transformation data set

a) All axes and the channel must be assigned to the same operating mode group.
b) The transformation Option must be available.
c) The transformation data are taken over internally by the control at restart (warm restart).Definition errors cause Alarm 3087 to be output. The incorrect machine data is coded in the block number in the alarm text.
d) NC MD 564* bit 6 must be set for all fictitious axes.
e) The axis names of the real and fictitious axes of a transformation data set should not be repeated. 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
8 = 1000	W = 1000
9 = 1001	Q = 1001
10 = 1010	E = 1010
11 = 1011	
12 = 1100	
13 = 1101	
14 = 1110	
15 = 1111	

Example: 0000 0010=Z
0001 1001=Q1

Legal names for axes, angles, chamfer and radius

A	*unassigned address*	**N**	sub block
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	feed	**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*

11.10.4 Transformation parameters

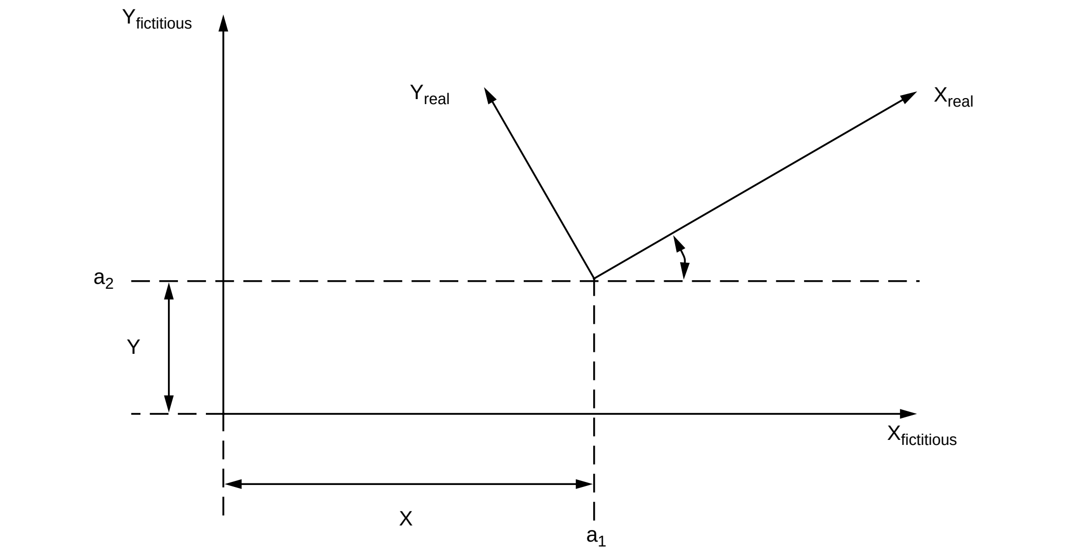
NC MD	Significance		
730	**1st transformation, parameter 1**		
731	**1st transformation, parameter 2**		
732	**1st transformation, parameter 3**		
733	**1st transformation, parameter 4**		
734	**1st transformation, parameter 5**		
735	**1st transformation, parameter 6**		
736	**1st transformation, parameter 7**		
737	**1st transformation, parameter 8**		
738	**1st transformation, parameter 9**		
739	**1st transformation, parameter 10**		
Sign	Input limits	Standard value	Units
+/-	99 999 999	0	units (IS)

NC MD 740 to 749 2nd, transformation, parameter 1 to 10
NC MD 750 to 759 3rd, transformation, parameter 1 to 10
NC MD 760 to 769 4th, transformation, parameter 1 to 10
NC MD 770 to 779 5th, transformation, parameter 1 to 10
NC MD 780 to 789 6th, transformation, parameter 1 to 10
NC MD 790 to 799 7th, transformation, parameter 1 to 10
NC MD 800 to 809 8th, transformation, parameter 1 to 10

The transformation parameters are needed for the 2D/3D coordinate transformation (and are of no consequence for the TRANSMIT coordinate transformation).

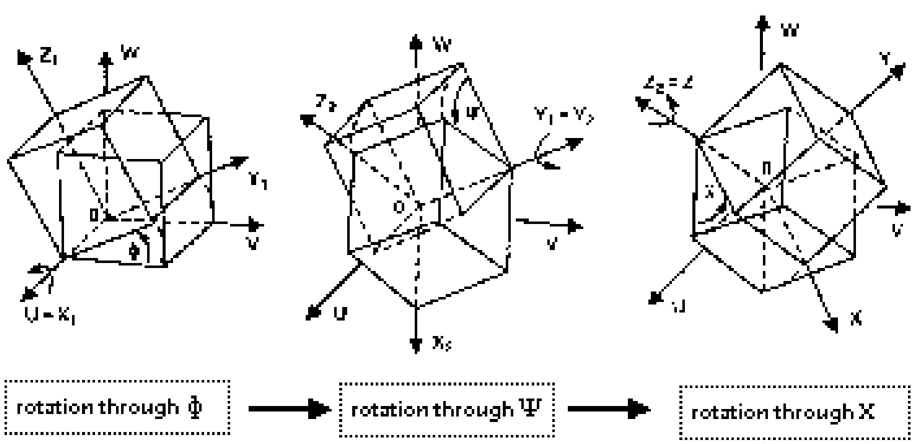
Transformation parameters for 2D coordinate transformation

Parameter 1: X shift of the real system in direction X relative to the fictitious origin a1 [unit: units (IS)].
Parameter 2: Y shift of the real system in direction Y relative to the fictitious origin a2 [unit: units (IS)].
Parameter 4: Angle of rotation of the real system relative to the fictitious system [unit: 10^{-5} degrees].
Parameter 10: Axis number which is used to calculate the G96 (constant cutting speed).



Transformation parameters for 3D coordinate transformation

Parameter 1:	X shift of the real system in direction X relative to the fictitious system [unit: units (IS)].
Parameter 2:	Y shift of the real system in direction Y relative to the fictitious system [unit: units (IS)].
Parameter 3:	Z shift of the real system in direction Z relative to the fictitious system [unit: units (IS)].
Parameter 4:	Angle of rotation , which occurs when the real coordinate system is rotated about the X axis (unit: 10^{-5} degrees].
Parameter 5:	Angle of rotation , which occurs when the real coordinate system is rotated about the Y axis (unit: 10^{-5} degrees].
Parameter 6:	Angle of rotation X, which occurs when the real coordinate system is rotated about the Z axis (unit: 10^{-5} degrees].
Parameter 10:	Axis number for axis which is used to calculate G96 (constant cutting speed).



X_n, Y_n, Z_n = rotated coordinates
X, Y, Z = real coordinate system
U, V, W = fictitious coordinate system
, , x = angle of rotation

11.10.5 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 (minimum acceleration value of A_{1R} to A_{5R}).
MD 280*		Maximum speed
MD 288*		JOG speed
MD 292*		Rapid traverse in JOG The speeds can be freely selected, as they are monitored by the control.
MD 304*		IPO parameter
MD 360*		Operating mode group of the axis
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 is therefore meaningless.
MD 564*	Bit 7	Axis exists
MD 568*		Encoding of the axis name
MD 576*		Axis not permitted in channel

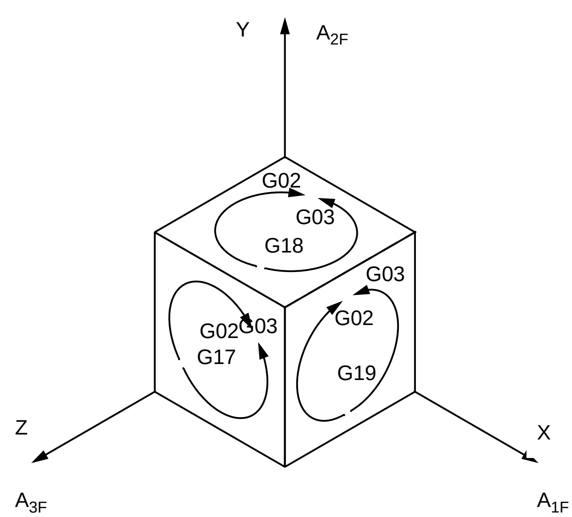
11.10.6 NC PLC interface signals

- In the case of fictitious axes, only the signals "JOG, rapid traverse overlay and handwheel 1, 2 and the input interface" are processed.
  The output interface is not accounted for. The signal "Reference point reached" is permanently set to 1.
- The program including the transformation is not stopped if the signal "Block axis" is set for a real axis in the transformation grouping. In this way an offset occurs between the transformation and the position control which can only be eliminated by selecting transformation.
- If a real axis in the transformation grouping is switched into control from the follow-up mode, an inverse transformation of the fictitious coordinates occurs automatically.
- If a real axis in the transformation grouping is occupied by "Feed stop", this applies to the entire grouping.
- When "Delete distance to go" occurs, the fictitious residual paths are cleared.
- The signal "Transformation active" is set in the channel-specific interface NC PLC for every channel in which the transformation is active (see also Interface Description).
- Warm restart is possible when transformation is selected if the transformation grouping does not change operating mode groups.
- Command channel
  In order to traverse a fictitious axis in JOG mode, the relevant transformation must first be activated in the channel assigned via machine data. The activation, i.e. the selection/ cancellation, of the transformation is a function mode of the command channel. The selection of coordinate transformation is described in the Interface Description, Part 1.

11.10.7 Explanation of the programming and operation of coordinate transformation

- Fictitious axes must not be programmed in the reset position (G130, G230, G330)
  Alarm 2043.
- A transformation may only be activated from the reset position, i.e. transition to a different transformation is only possible via a previous cancel block.
- The selection of coordinate transformation occurs as follows:
  G131 or G231 or G331 TRANSMIT
  G133 or G233 or G333 2D coordinate transformation
  G135 or G235 or G335 3D coordinate transformation

  The selection block must not contain any traversing movements, auxiliary functions etc.
- When transformation is selected, none of the real axes of the transformation grouping must be programmed 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.
- The offset of the cutter/tool nose radius compensation must be cancelled before activating the transformation (dependent on @714).
- Only **one** transformation can ever be selected in any one channel.
- Transformations which are running parallel in different channels must not refer to the same axes Alarm 2190.
- The plane definitions which are laid down in channel-specific machine data are valid for the real system. When a transformation is selected a plane is adjusted for the **fictitious** system. The fictitious plane is defined in the transformation data by assigning the fictitious axes. Deviations from the basic planes can be explicitly programmed via G16. In this context, fictitious axes must be given an equals sign, e.g. G16 Y1 = Z Z X1 =, where axes X1 and Y1 are the fictitious axes belonging to X and Y.The basic plane position is defined as G17 (A_{1F}-A_{2F}) with TRANSMIT.
  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.



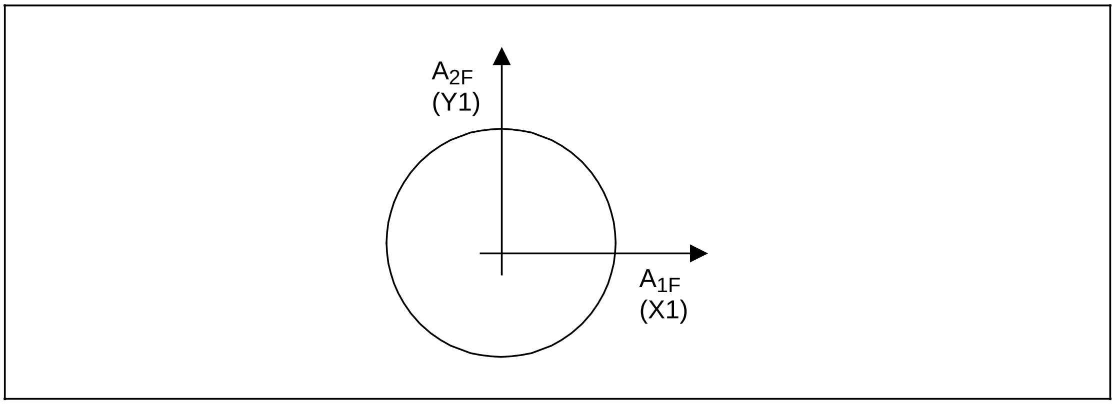
- A transformation must not be selected or cancelled within a contour block sequence.
- A block search to a part program where transformation is active is permitted.
- The automatic block search to a part program where transformation is active is **not** permitted.
- PRESET shifts of real axes are ignored in the case of transformation.
- Fictitious axes cannot be traversed with the handwheel. Real axes can only be traversed with the handwheel when transformation is not active.
- DRF is not possible in the case of fictitious axes.
- DRF is only possible in the case of real axes when a program block is active (not for NC STOP).
- The adjustable angle of rotation for coordinate system rotation (G54 to G57) must always be zero, from selecting trough to deselecting the coordinate transformation.
- The programmable angle of rotation for coordinate system rotation (G58, G59) must also be zero when transformation is being selected or cancelled.
- With G41**/**G42 the feedrate is restricted according to distance from the cutter centre to the transformation centre. The transformation centre cannot be approached with the cutter even if the programmed contour does not violate the centre.

11.10.8 Examples of coordinate transformation

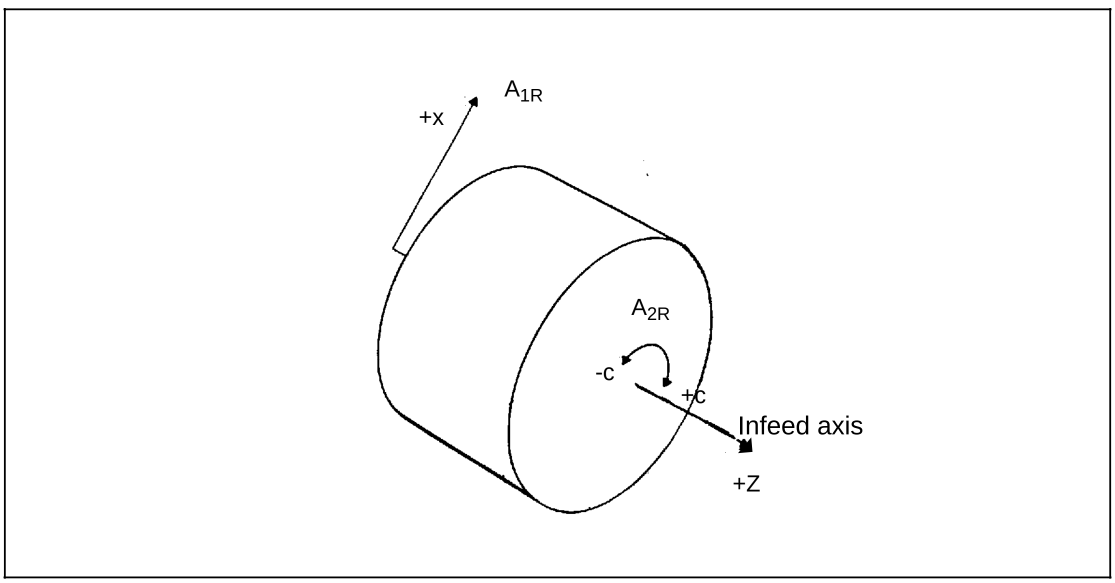
11.10.8.1 Example of the coordinate transformation TRANSMIT

A transformation data set for the TRANSMIT transformation must be defined as follows:

NC MD 5060	Channel number of the transformation Example 00000010 (binary form) (channel 2)		
NC MD 5061	G function for transformation selection	G131 G231 G331	0001 0001 0010 0001 0011 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 the infeed axis (real axis)	Example: 0000 0010 (Z)
NC MD 5065	Axis name of the 1st, real axis (A_{1R}) - linear axis	
NC MD 5066	Axis name of the 2nd real axis (A_{2R}) - rotary axis	



NC MD 5067 to 5069 unassigned (Input: 1111 1111), other entries can lead to malfunctions.

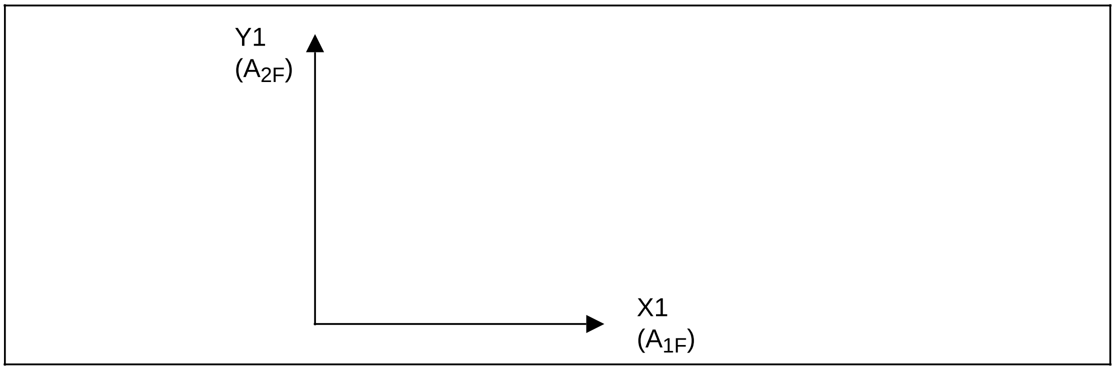
See Programming Guide for programming example.

11.10.8.2 Example of 2D coordinate transformation

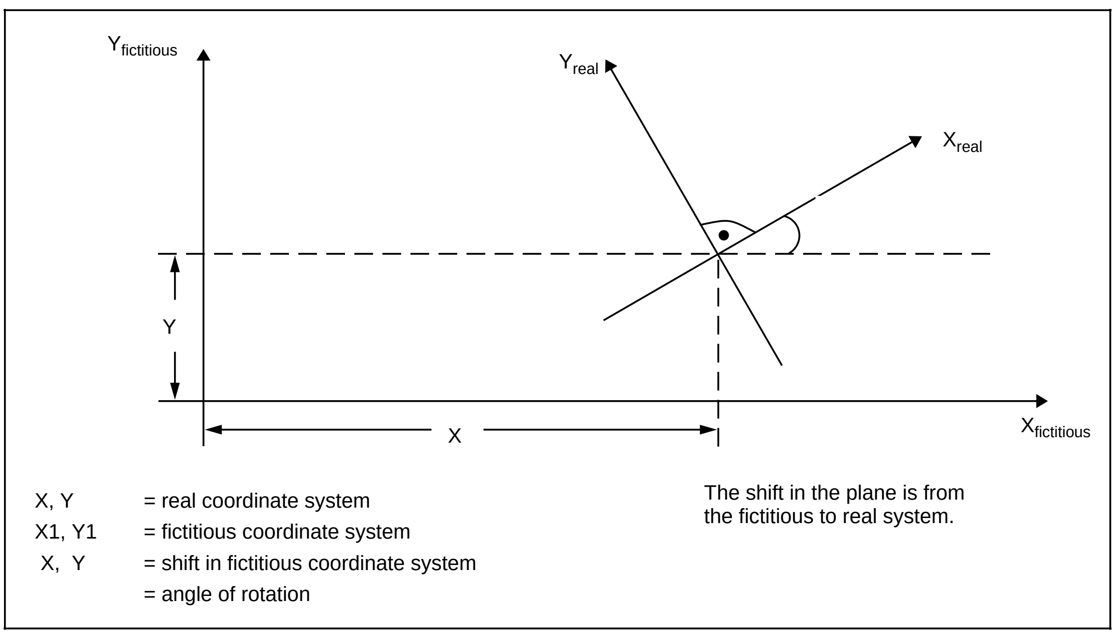
A transformation data for 2D coordinate transformation must be defined as follows:

NC MD 5060	Channel number of the transformation Example: 0000 0010 (binary form) (channel 2)		
NC MD 5061	G Function for transformation selection	G133 G233 G333	0001 0011 0010 0011 0011 0011

NC MD 5062	Axis name 1st, fictitious axis (A_{1F}) - linear axis	Example: 0001 0000 (X1)
NC MD 5063	Axis name 1st, fictitious axis (A_{2F}) - linear axis	Example: 0001 0001 (Y1)



NC MD 5064	Axis name of the 2nd fictitious axis again	Example: 0001 0001 (Y1)
NC MD 5065	Axis name of the 1st, real axis (A_{1R}) - linear axis	Example: 0000 0000 (X)
NC MD 5066	Axis name of the 2nd, real axis (A_{2R}) - linear axis	Example: 0000 0001 (Y)



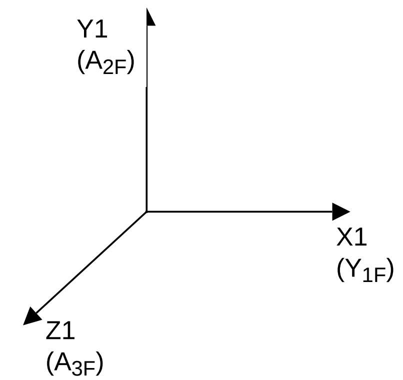
NC MD 5067 to 5069 unassigned (Input: 1111 1111), other entries can lead to malfunctions.

See Programming Guide for programming example.

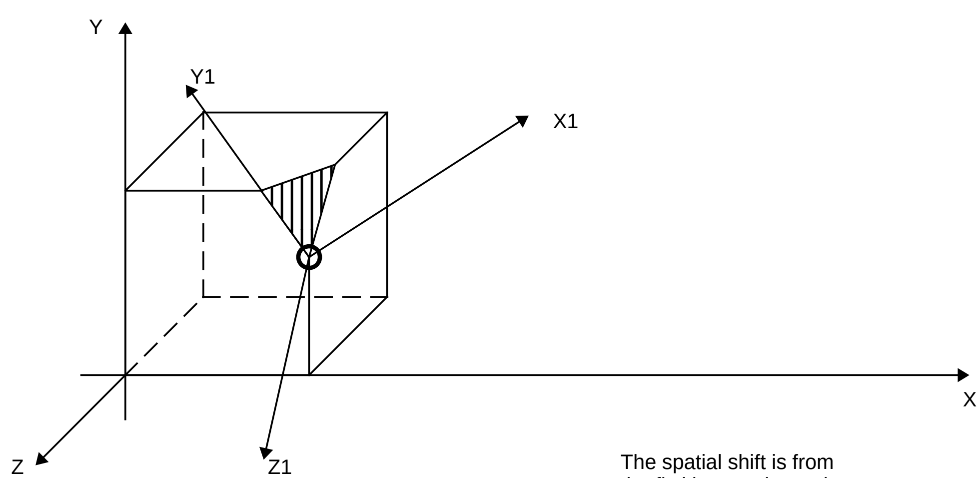
11.10.8.3 Example of 3D coordinate transformation

A transformation data set for 3D coordinate transformation must be defined as follows:

NC MD 5060	Channel number of the transformation Example : 0000 0010 (binary form) (channel 2)		
NC MD 5061	G Function for transformation selection	G135 G235 G335	0001 0101 0010 0101 0011 0101
NC MD 5062	Axis name 1st, fictitious axis (A_{1F}) - linear axis		Example: 0001 0000 (X1)
NC MD 5063	Axis name 2nd, fictitious axis (A_{2F}) - linear axis		Example: 0001 0001 (Y1)
NC MD 5064	Axis name 3rd, fictitious axis (A_{3F}) - linear axis		Example: 0001 0010 (Z1)



NC MD 5065	Axis name 1st, real axis (A_{1R})	- X
NC MD 5066	Axis name 2nd, real axis (A_{2R})	- Y
NC MD 5067	Axis name 3rd, real axis (A_{3R})	- Z



The spatial shift is from the fictitious to the real system.

X, Y = real coordinate system
X1, Y1, Z1 = fictitious coordinate system
X, Y, Z = to be input into the fictitious coordinate system

NC MD 5068 to 5069 unassigned (input: 1111 1111), other entries can lead to malfunctions.

See Programming Guide for programming example.

11.11 Extended thread package

The EXTENDED THREAD PACKAGE function is an Option.

The EXTENDED THREAD PACKAGE function comprises several individual functions:

- Following error compensation
- Multiple thread
- Thread recutting

To ensure correct operation of the function, make sure when setting up the control/function that the product of the interpolation time and the K_V factor of the thread cutting axis is not lower than a specified minimum value. The following table shows the minimum K_V factors derived from the preselected interpolation times.

Interpolation time (NC MD 155)	Minimum K_V (NC MD 252*)
2	3125
4	1562
6	1041
8	781
10	625
12	520
14	446
16	390
18	347
20	312

11.11.1 Following error compensation with thread cutting

This function compensates the following error of the cutting axis, which depends on the spindle speed in the case of thread cutting.

In this context the starting angle is corrected by the following error calculated from the spindle speed and the revolutional feedrate.

This allows a thread to be cut in several passes at different revolutional speeds without following error offset. However, this compensation only functions in the case of threads with constant pitch. The function must be selected with G37.

Prerequisites:

- The EXTENDED THREAD PACKAGE Option must be available.
- The K_V factors must be the same for all the axes involved in the thread cutting.

Activating the function

If all the prerequisites are fulfilled, no further action is required. The spindle speed must be constant within a cut because deviations from the ideal characteristic are unavoidable on account of the given dynamic response. If compensation is not possible, alarm 3089 "K_V factor for thread too small" is output.

11.11.2 Multiple thread

This function facilitates cutting a multiple thread with constant pitch.

Prerequisites:

The EXTENDED THREAD PACKAGE Option must be available.

Function description

When cutting a multiple thread, a separate starting angle must be programmed directly for each turn with G92 A.. in the part program. The starting angle refers to the spindle encoder zero mark (including zero mark offsets) and can be input as a value between 0 and 359.999 degrees. Angles above 360 degrees or below 0 degrees are calculated modulo 360.

The last angle input is retained with Power Off. The starting angle can also be entered directly in setting data 204* of the spindle data or programmed via R parameters.

Example: triple thread (offset by 120 degrees in each case)

G01 G90 X50 S200 M3 LF	Starting point
G92 A0 LF	First turn (0 degrees)
X100 LF	
:	
X50 LF	Starting point
G92 A120 LF	Second turn (120 degrees)
X100 LF	
:	
X50 LF	Starting point
G92 A240 LF	Third turn (240 degrees)
X100 LF	
:	

The function can be activated with G33 or G37. The speed for the individual steps can be modified only with G37.

11.11.3 Setting up thread recutting

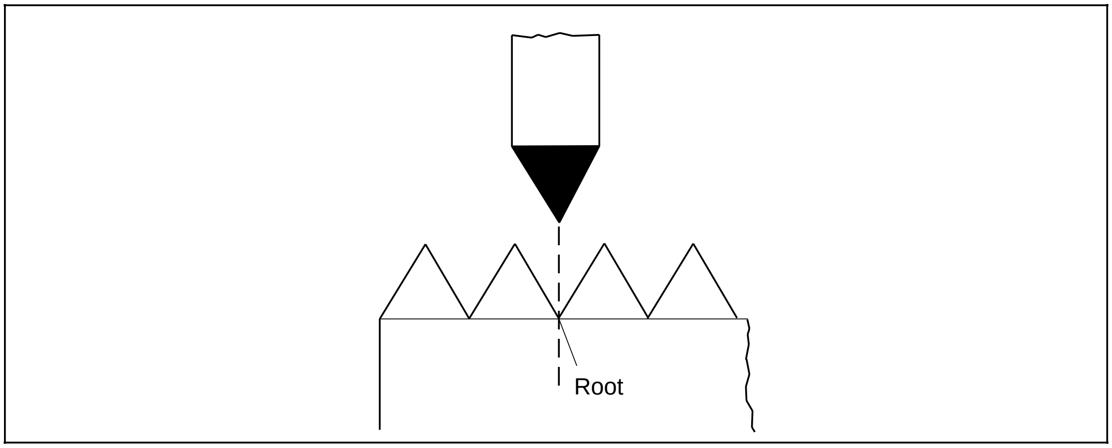
With this function, it is possible to recut an existing thread with constant pitch (G37) on a workpiece.

Prerequisite:

The EXTENDED THREAD PACKAGE Option must be available.

Function description

The workpiece with the thread must be clamped. The tool tip must be positioned in the JOG/INC mode so that it is precisely above the root (any position on the thread).



Press the "RECUT THREAD" softkey to obtain the "SETUP THREAD CUTTING" display.

The offset angle must be reported to the control here by pressing the "STORE POSIT." softkey. The previous offset angle and any starting angle are deleted. Correct execution of storing and calculation functions is indicated to the user by a tick after the start angle. The new offset angle is calculated and entered with the first thread cutting block of the part program.

The offset angle is a type of "spindle zero mark offset" made necessary because the workpiece has inevitably not been reclamped in the same angular position as when the thread was originally cut.

The offset angle deleted with Power On and when the "DEL. OFFSET" softkey is pressed.

11.12 Temperature compensation and interpolatory compensation

The functions TEMPERATURE COMPENSATION and INTERPOLATORY COMPENSATION are available as Options.

Corresponding data:

- IKA data 1 - 3 (IKA relations, error curve pointers, compensation points)
- NC MD 356* Monitoring max. IKA compensation values
- DB32 DRn Bit 6 IKA warning threshold
- TEMPERATURE COMPENSATION (H58) and/or INTERPOLATORY COMPENSATION (H57) Option must be available.

Description

Because of the increasing demand for accuracy with machine tools, it is necessary to introduce improved functionality as regards compensation for error factors between the machine and measuring system.

The acquisition of the actual value by the control is subject to errors owing to the following influences, for example:

- Temperature (thermal expansion)
- Mutual mechanical influences of axes (e.g. sag)
- Leadscrew error (spindle manufacturing tolerances)

These influences can be compensated within the control by means of temperature compensation or interpolatory compensation. The machine and measuring system parameters measured by laser when commissioning are stored in the control for utilization in the compensation calculation. The calculated absolute compensation values are integrated in the position control loop so that the axes approach the machine setpoint actually required.

A compensation facility is already available for compensating leadscrew errors (leadscrew error compensation, see Section 11.4), but is only of restricted use owing to the fixed grid spacing.

The new function contains two different compensation facilities which can be active at the same time.

- Temperature compensation (TC) to compensate thermal expansion
- Interpolatory compensation (IKA), e.g. for leadscrew error compensation, sag compensation with telescopic axes etc.

The function is active in all modes after the reference point approach of the basic and compensation axes. If an axis that has already been referenced is referenced again, the compensation is switched off after the cam has been triggered until the reference point approach is completed.

The resulting compensation amount is displayed in the service display in units of position control resolution.

The compensation axis must not be a fictitious axis.

The calculated compensation value (sum of TC and all IKAs) is included as an absolute position error in the actual value path calculation. The change in this value per IPO pulse must not exceed a given threshold. This threshold is distributed over several IPO pulses by means of a ramp function. The maximum permissible absolute value change, that is the maximum compensation speed, is equivalent to the incremental velocity (MD 300*).

Example:

MD 300* = 500 mm/min, IPO pulse = 20 ms
 maximum change in compensation value per IPO pulse = 166 µm.

IKA warning threshold

In the case of large compensation values, unexpected machine movements can occur which can be only partly restricted by the monitors. The current IKA compensation value is therefore checked according to the threshold value stated in MD 356*. If the value is exceeded, an axis-specific PLC interface signal is set (DB 32, DRn, bit 6). With reference to this signal the PLC can activate follow-up mode, for example, to halt the axis.

The maximum permissible compensation value is 9999 9999. The IKA is thus able to generate large traversing paths at the machine with the selection/deselection of the compensation or with modifications to the IKA relations.

By specifying geometry functions in the correction tables with the @40c function, complex geometric relations that extend beyond the range of normal programming can be implemented with the part program. This function can also implement collision avoidance, variable zero offsets etc..

In such instances larger traversing paths must be anticipated, whereas the distances traversed in the case of temperature, leadscrew error or sag compensation are in the range 0.01 to 0.2 mm. This discrepancy gives rise to two different strategies for monitoring the compensation movements with reference to the software position switches or movement abort conditions, for example. These must be realized largely in the PLC.

In order to ensure that unintentionally large IKA movements do not occur as a result of incorrect inputs (password protected), an IKA compensation value warning threshold has been implemented. Once this threshold is exceeded, the PLC can decide which measures are to be taken. To make certain that the contour errors which occur with large correction movements by the IKA are kept small, the speed must be as high as possible. The axial machine data "incremental velocity" (MD 300*) is also used in this context. The compensation axis executes the compensation movements at this speed.

The compensation axis always completes compensation movements, even if RESET is issued. Hardware limit switches, EMERGENCY STOP etc. apply as usual, also to the compensation axis.

The effective speed of an axis movement results from the partial setpoint and the compensation value change.

The alarm monitors (e.g. "set speed alarm threshold") always evaluate the effective set speed.

Position switch monitoring

When compensation is active, the limit switch monitors always evaluate the effective machine position:

NC setpoint - compensation value.

The deceleration ramp does not come into effect until the software limit switch is reached so that it can be overrun slightly, depending on the feedrate.

When a software limit switch is triggered, alarm 148* "software limit switch +" or alarm 152* "software limit switch -" is always output. If the software limit switch is reached with a compensation value active, the following new alarms are also output:

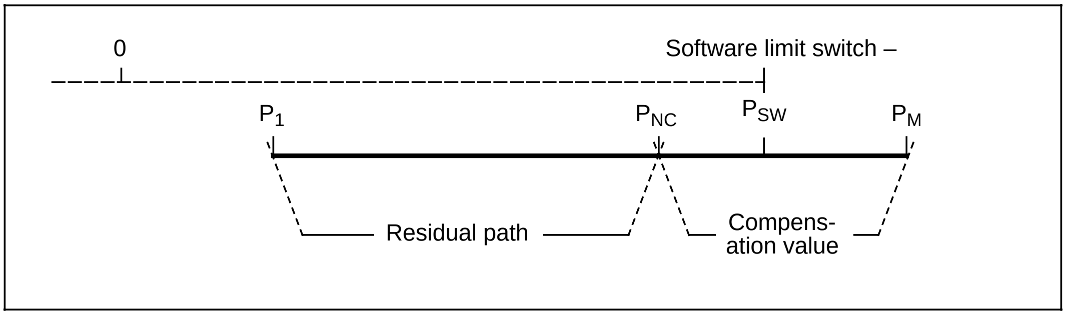
Alarm 1232* "Compensation limit switch +" or
Alarm 1236* "Compensation limit switch –"

Furthermore, the monitoring behaviour depends on the following factors:

Automatic/MDI mode:

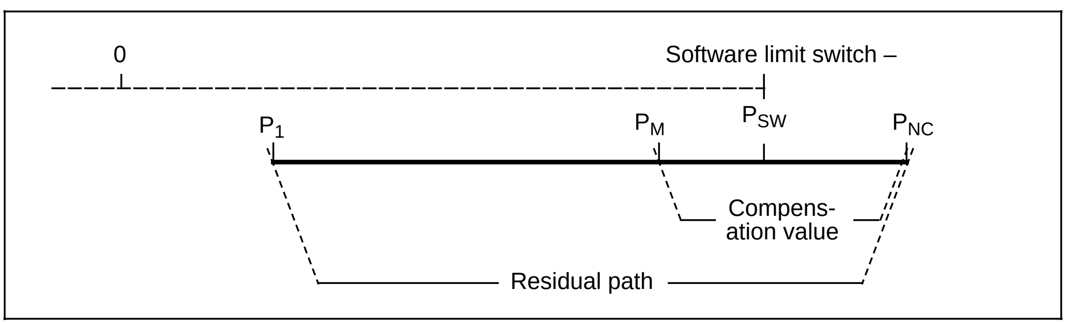
Before traversing, a part program block is always examined to check whether the limit range is reached. With compensation active, however, the resulting path can no longer be predicted at this stage. The following cases can occur when traversing according to this part program block:

a) With an NC setpoint P_{NC}, machine position P_M (compensation value negative) is obtained while traversing because of the compensation.



- Compensation movement in positive direction
- Software limit switch monitor before traversing only recognizes P_{NC} and does not, therefore, generate an alarm
- Alarm 148* generated during traversing when P_{SW} is reached.

b) With an NC setpoint P_{NC}, machine position P_M (compensation value positive) is obtained while traversing because of the compensation.



- Compensation movement in negative direction
- Software limit switch monitor before traversing only recognizes P_{NC} and thus issues alarm 2065 "programmed position behind software limit switch" although the machine position P_M that belongs to this NC setpoint is in front of the software limit switch.

JOG mode:

When traversing in the JOG mode, a residual path is always traversed from the current position P_1 to the software limit switch P_{SW}, so that the axis comes to rest precisely at the software limit switch.

With compensation active, the residual path is always calculated from the current machine position (NC setpoint – compensation value) to the software limit switch.

Two cases can arise depending on the error curve path up to the software limit switch:

a) Calculated residual path > residual distance according to machine

When a machine position P_{SW} is reached, alarm 148* is issued; the software limit switch is slightly overrun because of the deceleration ramp;

b) Calculated residual path < residual distance according to machine

The axis comes to rest before the software limit switch, so that the travel commands might have to be repeated many times in order to reach the software limit switch.

Remedy:

The calculation of the residual path to the software limit switch can be switched off for the JOG mode with MD 572*.5 = 1, that is to say the residual path is set to "infinity". When the software limit switch is reached, alarm 148* or 152* is output with cyclic position monitoring. The limit switch is reached directly in a single step and can be slightly overrun where applicable.

Interlocks:

In cases of axis-specific interlocking of compensation movements, the current compensation value is "frozen", that is to say it remains statically present. In this connection a distinction is made between:

- Direction-dependent interlocking
  - Software limit switches:

    After alarms 148* or 152* "software limit switch", or 1232* or 1236* "compensation limit switch" have triggered, both normal IPO movements and compensation movements in this direction are interlocked. In addition, all the channels of the mode group that is assigned to this axis are aborted.
  - Hardware limit switches
- Direction-independent interlocking:
  - Parking axis
  - Axis disable
  - Servo disable
  - Follow-up mode

All interlocks are cancelled once the cause itself is no longer present without waiting for RESET.

If the compensation axis is to be disabled by the PLC program or if the compensation movement is to be halted, this must take place via the axial PLC signals "servo disable", "follow-up mode", "axis disable" and "parking axis".

Axial feed hold and the channel-specific feed hold and override zero act on all axes of the mode group only by way of the leading axis or the channel.

Where a compensation axis is to be compensated by a basic axis via an IKA relation (see Section 11.12.2.2) and this basic axis loses its reference point (e.g. owing to parking axis), this compensation value component is not "frozen" but reduced to zero.

Iterative compensation

The following effect must be taken into account as regards the influence of the compensation value (K value):

A positive compensation value, for example, calculated with reference to an error curve results in traversing the axis (in a negative direction).

Depending on the size of the K value, this results in a new machine position. The axis is thus located in an area in which a different K value was measured when the error was measured. The difference between this new K value and the one last calculated depends on the size of the last K value and the gradient of the error curve between these points.

In order to calculate the new K value, the last K value must be eliminated again for the actual position as the input of the error curve, in other words the actual value is, as it were, compensated. The effective K value is then obtained iteratively. Note the following illustrative example:

The reverse compensation is controlled by bit 3 of the relevant IKA relation.

Bit 3 = 0: No assimilation of compensation value (uncompensated actual value), i.e. K_n = f (NC position)

Bit 3 = 1: Reverse compensation of the K value (compensated actual value), i.e. K_n = 1 (NC position, K_{n-1})

To calculate a K value without oscillation, irregularities in the IKA error curves must be avoided, this means:

1. The start point and end point of an error curve must be outside the traversing area or
2. The K value at the start or end point of an error curve must be zero.

Influence of directional compensation - backlash compensation

If an error curve is defined with positive and negative compensation values, the compensation axis acts accordingly in different directions. This does not mean, however, that this change in direction is detected and evaluated by any other directional IKA relation or backlash compensation that is present. These types of compensation recognize the change in direction with reference to the partial setpoints.

Relationship TC/IKA

The two types of compensation are completely independent of each another. The axis-specific sum derived from the individual results is the absolute compensation value for an axis. If several compensations are to be calculated, they have a single common actual value as their input parameter (compensated or uncompensated). Within an IPO cycle any previously calculated temperature control value is not taken into account when calculating the IKA values.

MODULO treatment for rotary axes

Like the former leadscrew error compensation, the actual position of a rotary axis (MD 564*.5 = 1) is converted to a MODULO value before searching in the error curve with this basic actual position. NC MD 344* "MODULO value of a rotary axis" determines the MODULO conversion value.

11.12.1 Temperature compensation

The TEMPERATURE COMPENSATION function is available as an Option.

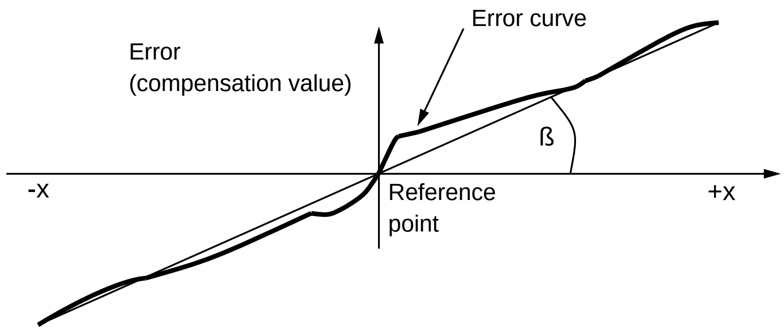
With this type of compensation the compensation values that are valid for the current temperature are transferred by the PLC to the NC via the command channel. The necessary compensation values must be calculated by the machine manufacturer in the PLC first.

11.12.1.1 Modification types

Two different types of temperature modification exist:

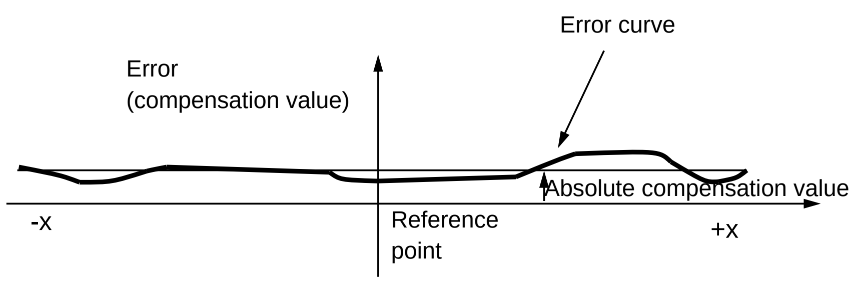
Position-dependent actual value modification

The temperature-related actual value deviation for an axis can be represented in the form of an error curve. It can be assumed that the position-dependent trace occurs uniformly from a reference point. It can be approximated by a straight line, the gradient of which depends on the temperature.

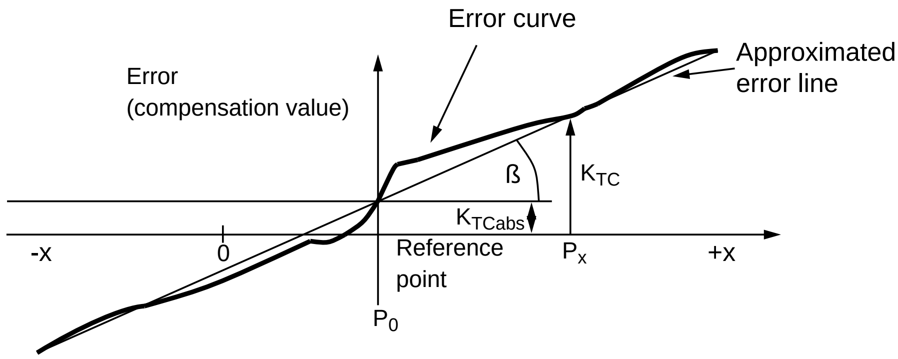


Position-independent actual value modification

The measured actual axis value deviates by an absolute temperature-dependent value, which is independent of the actual position of the axis. This error curve can also be approximated by a straight line.



The two error influences are superimposed additively so that the following approximate behaviour occurs with the actual value modification.



Where:

K_{TC} : Temperature compensation value for axis i at position P_x

K_{TCabs} : Position-independent temperature compensation value for axis i

P_x : Actual position of axis i

P_0 : Reference position for axis i

tanß : Coefficient for position-dependent temperature compensation (depends on the temperature and the coefficient of linear expansion of the machine).

$$\tan ß = \frac{K_{TC} - K_{TCabs}}{P_x - P_0}$$

Caution:

Reference point P_0 must not coincide with the zero point of the axis.

The approximated error line of the axis is now used by the control for the temperature compensation at this axis.

As the error line is valid only for the current temperature value, the parameters of the new error line must be transmitted to the NC again when the temperature rises or falls.

Only in this way can it be ensured that the thermal expansion is always correctly compensated.

11.12.1.2 Function description

An error curve can be defined for each axis. The parameters of this error curve are transferred via the command channel from the PLC to the NC.

The following parameters must be defined and transferred for each error curve:

- A position-independent compensation value KT_{Cabs}
- The reference point P_0 for the position-dependent compensation
- The coefficient tan ß of the position-dependent compensation
- The activation flags of the two compensation types

These values are computed internally with the current actual position in each case to obtain the compensation value K_{TC}. This compensation value is then taken into account in the position control.

The following applies to the sign:

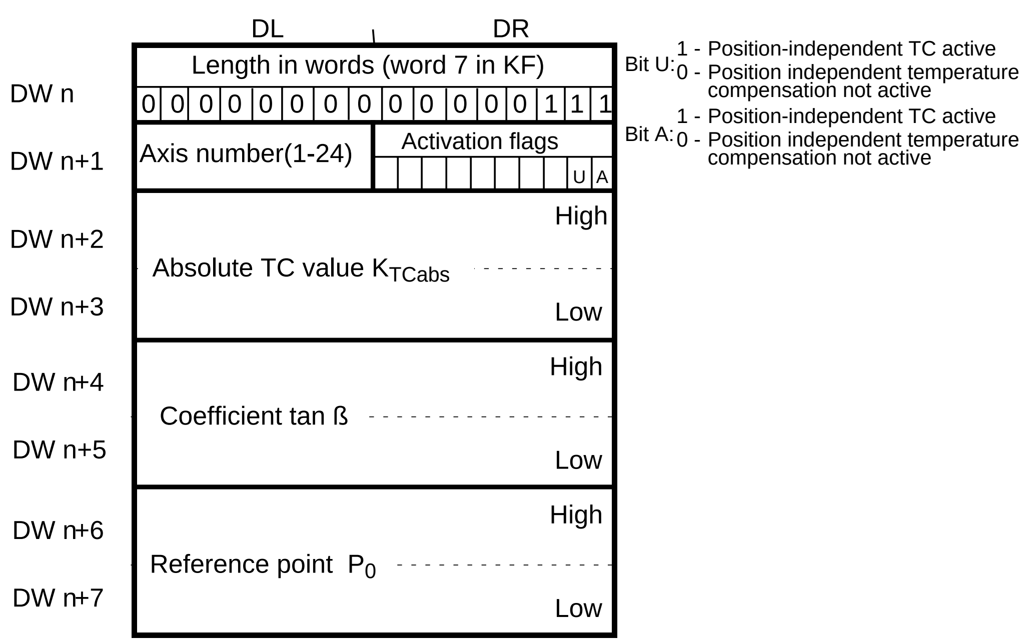
Error K_{TC} = machine position - setpoint position
i.e. with a positive K_{TC}, the axis moves in a negative direction.

11.12.1.3 Data structure

The transfer of the data from the PLC via the command channel to the NC is effected by function number 9 (entry in DB41).
The transfer must be made separately for each axis.

The content of the user DB is as follows for each axis:

Structure of user DB:



Activation flags:

These flags determine which TC process is to be used.

Data format of user DB

- Length in words: always value 7 in KF format
- Axis number: values 1 to 24 in KF format
- Absolute TC value K_{TCabs}: with sign in units (MS) in KF format (value range ±1072 000000)
- Coefficient tan ß: with sign with place value 2^{-31} in KF format (value range -1....+1)

  The following table illustrates the data format of tan ß as an example

ß	tan ß	(tan ß) · (2^{31}–1) (dec.)	(tan ß) · (2^{31}–1) (hex.)
0	0	0	0000:0000
30	0.577	1 239 850 262	49E6:9D16
45	1	2 147 483 647	7FFF:7FFF
-30	-0.577	-1 239 850 262	B619:62EA
-45	-1	-2 147 483 647	8000:0001

- Reference point P_0: with sign in units (MS) in KF format (machine reference system, value range ±1072 000000)

The values K_{TCabs}, tan ß and P_0 can be entered as the result of a floating-point calculation with subsequent fixed-point format conversion.

11.12.1.4 Activating the function

- The TEMPERATURE COMPENSATION Option must be available.
- Transfer of the currently valid compensation values via the command channel to the NC (cyclically, or depending on the NCDPLC interface). For a more detailed description of the command channel see Interface Description Part 1, Signals.

Caution:

Because the compensation values are immediately taken into account in the position control, the surface quality can be impaired in the case of a traversing axis if the change in the compensation amount between two data transfers is too great.

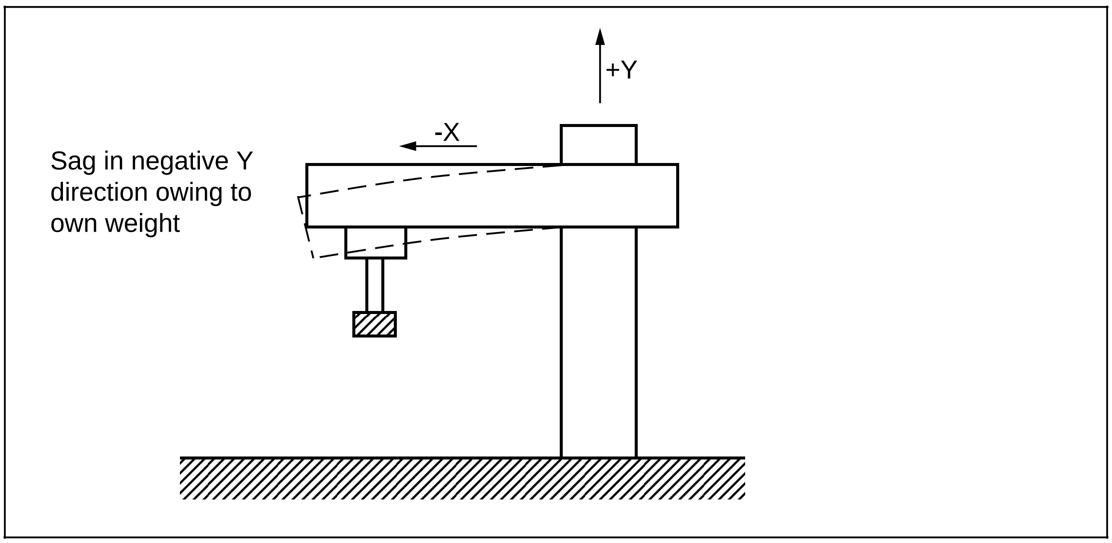
11.12.2 Interpolatory compensation

The INTERPOLATORY COMPENSATION function is available as an Option (not with export version).

The position of a (basic) axis can influence the absolute position of another (compensation) axis without this being noticed by the measuring system. This influence can be corrected by interpolatory compensation.

Example:

Cross-arm axis



The further the machining head travels in the negative X direction, the more the cross-arm axis sags in the negative Y direction.

The Y value must therefore be compensated as a function of the current X position.

11.12.2.1 Function description

A maximum of 32 mutually independent compensation relations can be defined for the interpolatory compensation (IKA). These compensation relations can all be active at the same time. A compensation relation describes the dependence of the compensation axis position on a basic axis position.

Each IKA relation is assigned an error curve. This curve describes the path of the deviations and comprises a sequence of actual positions (basic axis) and the associated compensation values (compensation axis).

In contrast to leadscrew error compensation, in this case the interpolation points (actual positions of the basic axis) can be entered with variable distances.

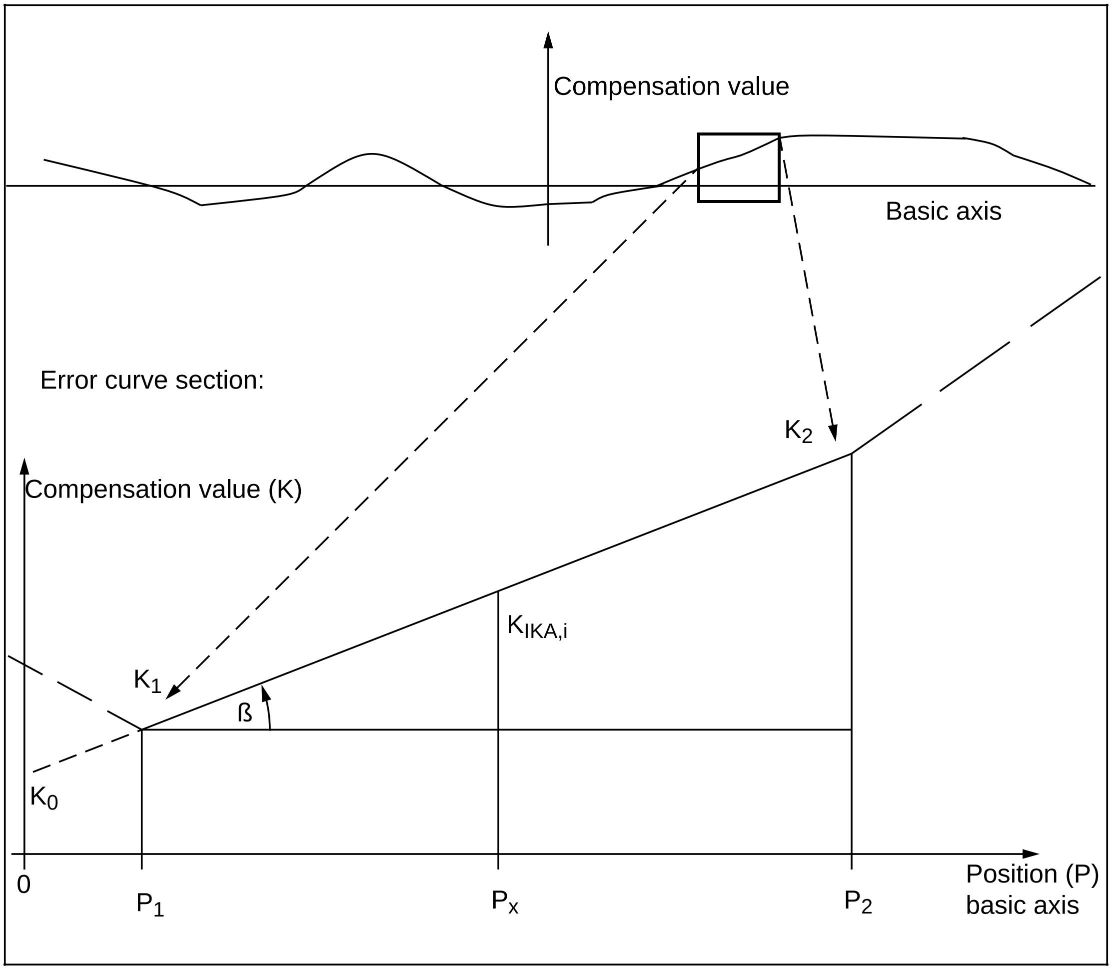
When used as leadscrew error compensation, the basic and compensation axes must be the same.

If the basic axis is located between two interpolation points, the compensation value is formed by interpolation from the external compensation values (interpolation points are connected by a path).

Note:

In the synchronous mode a C axis can be the basic axis of an IKA relation if the spindle is synchronized. Operation as a compensation axis is possible only in the C axis mode.

Complete path of error curve



As illustrated by the error curve, the absolute compensation value K_{IKA} is calculated within an error curve section P_1P_2 as:

$K_{IKA} = P \cdot \tan ß + K_0$, where $\tan ß = \dfrac{K_2 - K_1}{P_2 - P_1}$

The parameters $K_{IKA,i}$ and P_X are entered in the compensation data by the user. The values tan ß and K_0 are calculated automatically by the control on runup or on a warm restart, so that they are directly available during the compensation.

The compensation value K_{IKA} can also be modified by a compensation factor to include the influence of the workpiece weight where applicable without changing the error curve or in order to compensate different axis position control resolutions (e.g. basic axis 10^{-3} and compensation axis 10^{-4}):

$K_{IKAnew} = G * K_{IKAold}$ (Standard: G=1)

The compensation points of all (max. 32) error curves are entered in a list. The assignment of the compensation points to the IKA error curves is made with start and end pointers.

Influence of the reference point

Once specified, an error curve of an IKA relation can be displaced as required in the direction of the basic axis:

$K_{IKA} = G \cdot ((P - P_0) \cdot \tan ß + K_0)$

The reference point has the standard value 0.

11.12.2.2 Data structures and data assignment

There are three different data fields:

IKA relations

This data field contains the following data of all the available IKA relations:

Data type	Value range	Data type No. for command channel/RS232C (V.24)/ @ function
• Number of IKA relation	1...32	–
• Flag byte of IKA relation	–	0
• Relation active	0/1	11
• Direction-dependent	0/1	12
• Compensation direction	0 (neg. direction) /1 (pos. direction)	13
• Compensated actual value	0 (uncompensated) /1 (compensated)	16
• Error curve number	1...32	1
• Basic axis	1...24	2
• Compensation axis	1...24	3
• Compensation factor G (in 1/1000)	±16 000	4
• Reference point P0 (units MS)	± 99 999 999	15

Note:

The data types "relation active (B)", "direction-dependent (R)", "compensation direction (P)" and "compensated actual value (K)" can also be addressed globally as a byte under data type number 0 (format: 0000KPRB).

Error curve pointer

The compensation points are assigned to the error curves here. By stating the start pointer and the end pointer, a section from the COMPENSATION DATA data field is assigned to the error curve.

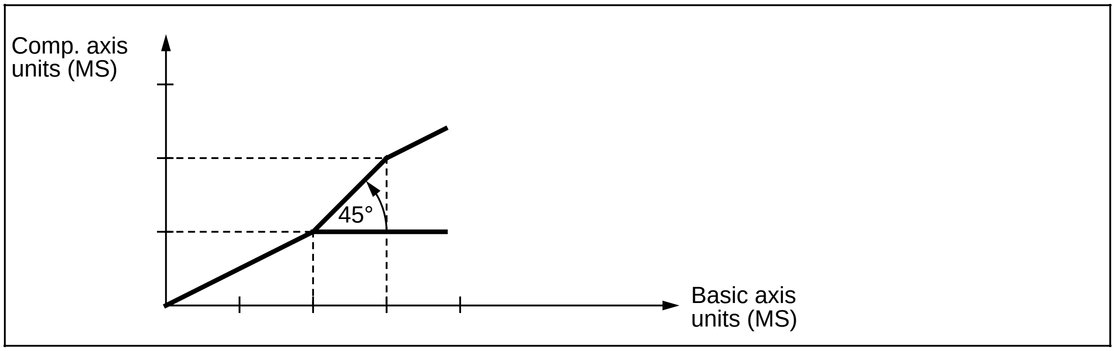
Data type	Value range	Data type No. for command channel
• Error curve number	1...32	1
• Start pointer	1...3000	5
• End pointer	1...3000	6

Compensation points

In this data field each compensation point is assigned a position value of the basic axis and an associated compensation value of the compensation axis. This field has 3000 compensation points.

Data type	Value range	Type No.
• Compensation point	1... 3000	
• Actual position (basic axis)	±99 999 999 (units MS)	7
• Compensation value of compensation axis	±99 999 999 (units MS)	8

With IKA compensation, the basic axis and the compensation axis can have different position control resolutions. Because the gradient of the error curve within a section must not be greater than 45°, the amount of compensation must not exceed the traversing distance of the basic axis (with equal position control resolutions).



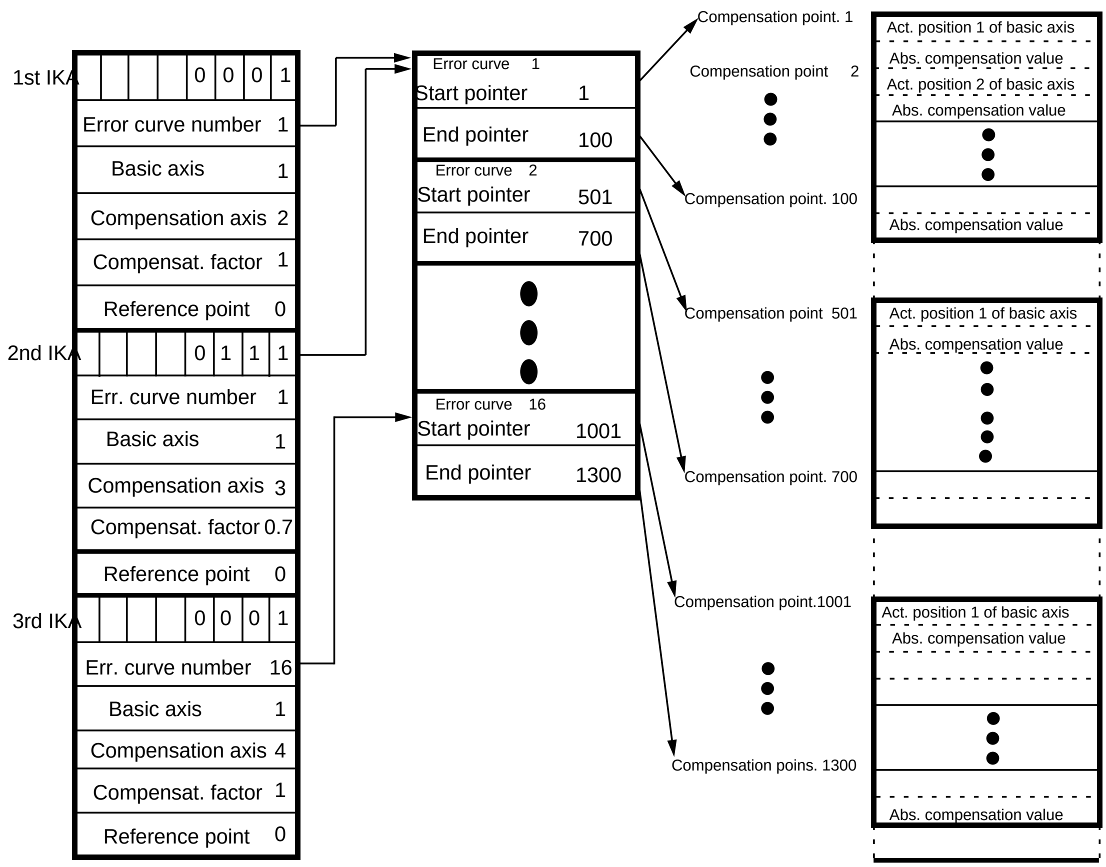
If the basic axis has a position control resolution of $0.5 \cdot 10^{-3}$ mm and the compensation axis a position control resolution of $0.5 \cdot 10^{-4}$ mm, for instance, this would result in a maximum gradient of $\tan^{-1}$ (0.1) = 5.71°.

A steeper gradient up to max. 45° can only be achieved in such a case by setting the compensation factor to 10 000 (10x) and entering the compensation values in units of 10^{-3} mm [units (MS) of basic axis].

On the other hand, steeper gradients (e.g. up to 84.3°) are possible if the position control resolution of the basic axis is $0.5 \cdot 10^{-4}$ mm and that of the compensation axis is $0.5 \cdot 10^{-3}$ mm.

Example for data assignment between the three data fields:

- A total of 3 IKA relations are to be active.
  The influences of axis 1 on axes 2, 3 and 4 are to be compensated.
- Compensation is to take place with two error curves (FK):
  FK1 is located between compensation point 1 and compensation point 100
  FK16 is located between compensation point 1001 and compensation point 1300
- The 2nd IKA also operates with error curve 1, but with a compensation factor of 0.7



Read and write access to these three data fields is as follows:

- Input via operator panel – IKA relations
  – Error curve pointers
  – Compensation points
- Via part program with @30C/@40C – IKA relations
  – Error curve pointers
  – Compensation points
- Via RS232C (V.24) interface (data in/out) – IKA relations
  – Error curve pointers
  – Compensation points
- By PLC via command chan. (function Nos. 10 and 11) – IKA relations

Note:

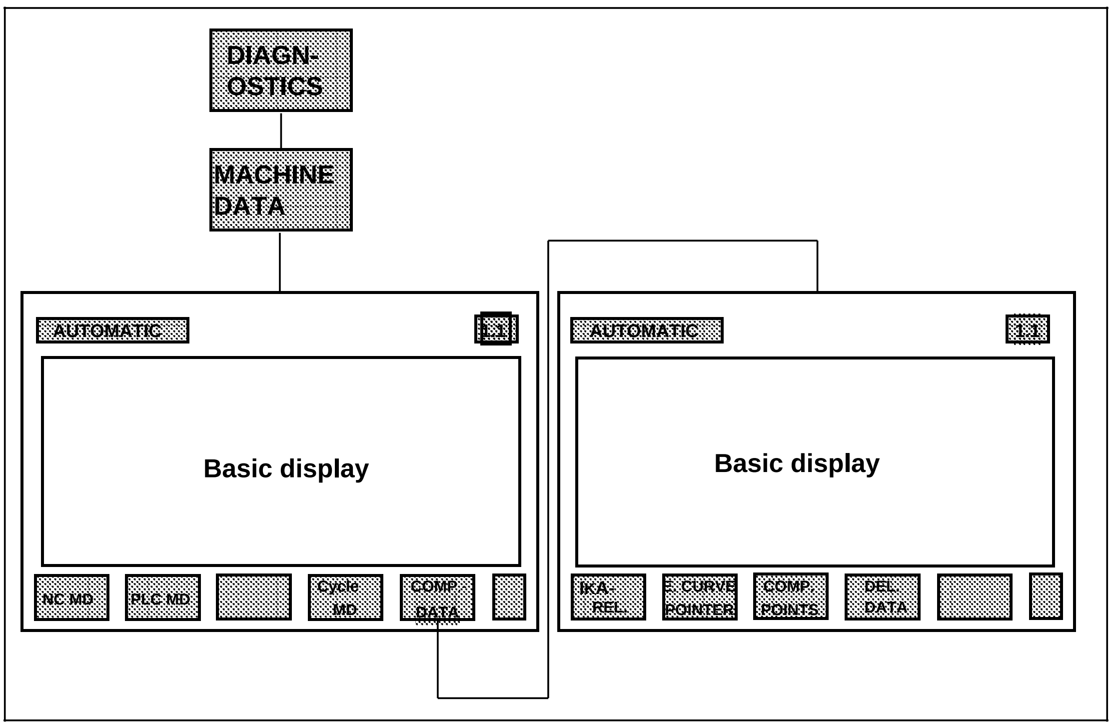
In the event of a value change, the data become active immediately. When the compensation points are edited, the error curves (see Section 11.12.2.1) are not converted until POWER ON or warm restart. Editing error curves that are active can cause traversing errors until the conversion is made and must therefore be avoided.

11.12.2.3 Activating the function

- The INTERPOLATORY COMPENSATION (H57) Option must be available.
- The three data fields must be entered correctly.
- POWER ON/warm restart for converting the error curves.

11.12.2.4 Data access via operator panel

The compensation data are entered with the aid of input displays. The same conventions apply as for the machine data (key word: interlocking). The compensation data can be deleted only in normal mode (not in start-up mode).



11.12.2.5 Data access via PLC (command channel)

The IKA relations are written and read via the command channel with the function numbers

- 10 (read NC data via command channel) and
- 11 (write NC data via command channel).

This function is initiated in the same way as the familiar command channel functions.

Read IKA data (command channel function 10)

As usual, a user DB is parameterized to initiate the function from the PLC. The numbers of the user DB and of the first data word "DWx" are entered in DB41 under "DW_{m+2}/DW_{m+3}".

User DB with data specification for function initiation:

	DL (bits 15 … 8)	DR (bits 7 … 0)
DW_X	Length in words	
DW_{X+1}	Channel number (=0)	Number of IKA relations
DW_{X+2}	Data group identifier (=03)	Data type (=0)
DW_{X+3}	Data number	
DW_{X+4}		OCB Offset data type/number
DW_{X+5}	Number format (=0)	Dimension identifier (=0)

The length in words refers only to the data specification (in this case 5).

In addition, the target DB with initial number "DWy" must be stated in DB41 under "DW_{m+4}/DW_{m+5}". The read data are then transferred to this location and have the following structure:

Target DB after function execution:

	DL	DR	
	15 14 13 12 11 10 9 8	7 6 5 4 3 2 1 0	
DW_{Y}	Length in words		
DW_{Y+1}	Flags	No. of error curve	
DW_{Y+2}	Compensation axis	Basic axis	
DW_{Y+3}	Compensation factor		
DW_{Y+4}	Reference point (High)		
DW_{Y+5}	Reference point (Low)		
DW_{Y+6}	Flags	No. of error curve	
DW_{Y+7}	Compensation axis	Basic axis	
DW_{Y+8}	Compensation factor		
DW_{Y+9}	Reference point (High)		Max. 4 freely selectable IKA relations
DW_{Y+10}	Reference point (Low)		
DW_{Y+11}	Flags	No. of error curve	
DW_{Y+12}	Compensation axis	Basic axis	
DW_{Y+13}	Compensation factor		
DW_{Y+14}	Reference point (High)		
DW_{Y+15}	Reference point (Low)		
DW_{Y+16}	Flags	No. of error curve	
DW_{Y+17}	Compensation axis	Basic axis	
DW_{Y+18}	Compensation factor		
DW_{Y+19}	Reference point (High)		
DW_{Y+20}	Reference point (Low)		

The length in words states the number of user data read-in (in this case max. 20).

Write IKA data (command channel function 11)

As usual, a user DB is parameterized to initiate the function from the PLC. The numbers of the user DB and of the first data word "DWx" are entered in DB41 under "DW_{m+2}/DW_{m+3}". The user data must be stored in the user DB directly after the data specification (DW_{x+1} to DW_{x+5}). The length in words refers to the entire range of the data specification and user data (max. 25).

User DB with data specification and write data:

	DL (bits 15–8)	DR (bits 7–0)	
DW_X	Length in words		
DW_{X+1}	Channel number (=0)	Number of IKA relations	
DW_{X+2}	Data group identifier (=03)	Data type (=0)	
DW_{X+3}	Data number		
DW_{X+4}	OCB	Offset data type/number	
DW_{X+5}	Number format (=0)	Dimension identifier (=0)	
DW_{X+6}	Flags	No. of error curve	Max. 4 freely selectable IKA relations
DW_{X+7}	Compensation axis	Basic axis	
DW_{X+8}	Compensation factor		
DW_{X+9}	Reference point (High)		
DW_{X+10}	Reference point (Low)		
DW_{X+11}	Flags	No. of error curve	
DW_{X+12}	Compensation axis	Basic axis	
DW_{X+13}	Compensation factor		
DW_{X+14}	Reference point (High)		
DW_{X+15}	Reference point (Low)		
DW_{X+16}	Flags	No. of error curve	
DW_{X+17}	Compensation axis	Basic axis	
DW_{X+18}	Compensation factor		
DW_{X+19}	Reference point (High)		
DW_{X+20}	Reference point (Low)		
DW_{X+21}	Flags	No. of error curve	
DW_{X+22}	Compensation axis	Basic axis	
DW_{X+23}	Compensation factor		
DW_{X+24}	Reference point (High)		
DW_{X+25}	Reference point (Low)		

Note:

If the command channel is initiated and no command channel error occurs, the data are always transferred even if the addressed axis is currently active.

Owing to sudden setpoint changes, this can lead to a reduction in workpiece surface quality. Data transfer has to be possible even with active axes, however, so that the compensation behaviour can be modified during long traversing blocks.

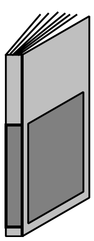
The user can execute a controlled data transfer by utilizing the axisDspecific NC/PLC interface.

Data format of user DB

Only certain data words are required with the read/write IKA data function. Those that are not required must be preset to "0".

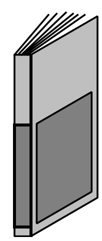
• Length in words	The significance of this parameter (enter in KF) depends on the function number.
– Read NC data (10):	User DB: 5 Target DB: Number of transferred words
– Write NC data (11):	5 + number of words to be written
• Number of IKA relations:	The number of IKA relations being transferred (KF) must be entered here
• Data group identifier	03_{HEX} must be entered here for the IKA data.
• Data number	The number of the first IKA relation whose data are being transferred must be entered here. Range: 1-32 in KF
• Offset data type/number	In the case of IKA data, the value "1" must always be entered here in KF.
• Offset control bit (OCB)	In the case of IKA data, the value "0" must always be entered here in KF.

The value range of the IKA relations is already given in Section 11.12.2.2. They must be entered in KF format.



The following publication contains more detailed information on the command channel function:

SINUMERIK 880 GA2 Order No. 6ZB5410-0HE02
Interface Description Part 1
Signals



11.12.2.6 Data access via @ function in part program

The IKA relations can be modified by means of the @ functions @30C and @40C.

- Read IKA relation:

Part program	@30C <Var> <Value 1> <Value 2>
WS800 syntax	<Var> = MDIKA <Value 1> <Value 2>

Var :	R parameter with read value
Value 1:	Constant with data type (see Section 11.12.2.2)
Value 2:	Constant with number of IKA relation

  Example:

  @30C R10 K2 K11 LF

  The basic axis No. (data type 2) of the 11th IKA relation is stored in R10.

- Write IKA relation:

Part program	@40C <Var> <Value 1> <Value 2>
WS800 syntax	MDIKA (<Value 1> . <Value 2>) = <Value>

Var:	R parameter with write value
Wert 1 :	Constant with data type (see Section 11.12.2.2)
Wert 2 :	Constant with number of IKA relation

  Example:

  @40C R20 K1 K3 LF

  The error curve No. (data type 1) of the 3rd IKA relation is transferred from R20 to the data field.

The writing of IKA relations with the aid of @40C can be disabled with NC MD 5012 bit 2.

Note:

The IKA relations cannot be read/written in the channels defined on the second NC CPU.

11.12.2.7 Data access via RS232C (V.24) interface

The IKA data are divided between three RS232C (V.24) data types, each of which can be read out/in separately. The conventions for reading in/out are the same as for the machine data except that in the set-up mode IKA data cannot be read in.

Output of IKA relations:

```
% IKA1
T xx N yyy = 00000101
          :
```

(xx: 00 IKA bits, 01 Error curve, 02 Basic axis, 03 Compensation axis,
04 Compensation factor, 15 Reference point)
(yyy: No. of IKA relation)

Output of error curve pointers:

```
% IKA2
T 05 N yyy = 100         T 06 N yyy = 129
T 05 N yyy = 130         T 06 N yyy = 2100
          :                        :
```

(05 Start pointer; 06 End pointer)
(yyy: No. of error curve)

Output of compensation points:

```
% IKA3
T 07 N yyy = 10000       T 08 N yyy = 12
T 07 N yyy = 10010       T 08 N yyy = 14
          :                        :
```

(07 Actual position; 08 Compensation value)
(yyy: No. of compensation point)

Example:

```
% IKA1
T  00 N    2 = 00000001          2nd IKA relation is active
T  01 N    2 = 6                 2nd IKA relation has error curve 6
T  02 N    5 = 1                 5th IKA relation has basic axis 1
T  03 N    5 = 2                 5th IKA relation has compensation axis 2
M02

% IKA3
T  07 N    110 = 0               110th compensation point has actual position 0
T  08 N    110 = 1000            110th compensation point has compensation value 1000
M02
```

The data output is controlled from the softkey menu bar:

IKA-DAT RELAT	:	Output of IKA relations (data type 0-4, 15)
IKA-DAT E.CURVE	:	Output of error curve pointers (data type 5-6)
IKA-DAT COMP-P.	:	Output of compensation points (data type 7-8)

The conventions are the same as those that apply with the familiar data. In the event of erroneous data contents, alarm 32 "Data format error RS232C (V.24)" is output.

11.13 Rapid block change with FIFO function

The RAPID BLOCK CHANGE WITH FIFO function is an Option.

11.13.1 Corresponding data

- The FIFO (**f**irst **i**n **f**irst **o**ut) Option must be available
- NC MD 558* bit 1 FIFO active in channel

11.13.2 Function description

When executing part programs with short traversing blocks sudden drops in the feedrate can occur in certain circumstances because the block preparation takes longer than block execution in such cases. The block change time (block preparation + block transfer) is too high in these instances.

Execution with the FIFO function results in a much shorter block change time.
This is achieved by a block buffer (FIFO memory) which is filled first after NC Start or by programming G171.
The next blocks of the part program are initially buffered in this FIFO memory in a prepared (pre-decoded) condition. The program is not initiated or execution is not resumed until until the memory with the prepared blocks is full.

If the NC happens to execute a block with a long traversing path or auxiliary functions, the time gained is used to refill the FIFO memory (The FIFO is filled in the background).
In the case of long part program sections with short traversing blocks, however, there might not be sufficient time available for refilling the FIFO memory. The complement of pre-decoded blocks in the FIFO memory can thus be exhausted and the problem of feedrate slumps can return.

Filling FIFO memory with G171

In order to avoid a drop in the feedrate in an important program section, the FIFO memory can be filled selectively prior to the block in question. Such selective filling is initiated by entering G171 in the part program. This enables the following program section to be executed from the FIFO memory with short block change times.

G171 has the following sequential effect:

- Interruption of execution
- Filling of FIFO memory
- Resumption of execution

Execution is interrupted until the FIFO memory is full, until a block with M02/M30 or @714, or until a block that generates a @714 (setting data block).

G171 is useful only in the AUTOMATIC mode.

Behaviour with "program hold"

If the part program is interrupted and edited during execution, the blocks already contained in the FIFO memory are executed in their original condition.

Note:

If TO or ZO values are edited in the stop state, the changes are also included in the blocks already contained in the FIFO memory.

FIFO memory size

The FIFO memory available to each NC CPU can buffer up to 60 blocks. This total memory capacity can be distributed between the individual channels with NC MD 558* bit 1.

The following relationship is valid:

FIFO active in one channel:	60 FIFO blocks available in channel
FIFO active in two channels:	30 FIFO blocks available in channel

The FIFO function cannot be activated in more than 2 channels of an NC CPU.

Any activation of a FIFO buffer for a channel that does not exist is ignored.

11.14 2nd/3rd operator panel for SINUMERIK 880 GA2

The 2ND/3RD OPERATOR PANEL function is an Option.

11.14.1 Corresponding data

The "2nd/3rd operator panel" Option must be available.

NC MD 5030	Declaration of operator panel
NC MD 5031	Operator panel configuration display
NC MD 5183 to 5184	Operator panel switchover active
PLC MD 6026 bit 4	Transfer of customer keys
DB40 DW43 bit 8-10	Activation of operator panel
DB40 DW43 bit 0-2	Active operator panel message
FY20 bit 0-2	"Operator panel ready" signal.

11.14.2 Function description

With the SINUMERIK 880 GA2 a 2nd and 3rd operator panel can be connected to the first one as operator terminals for tool loading stations, pallet loading/unloading stations etc. .

All the operator panels (max. 3) have the same scope of functions.

The monitors and LEDs of all 3 operator panels have the same display contents and status.

Only the keyboard of the currently active operator panel is operational.

The freely assignable customer keys are active at the same time on all operator panels so that they can be used to switch over to one of the other operator panels (selection with NC MD 5183 to NC MD 5184). The selected customer keys are ored by the system.

A customer panel is activated with the interfaces DB40 DW43 bits 8 to 10.

The message indicating which operator panel is currently active is issued by DB40 DW43 bits 0 to 2.

The maximum distance between two operator panels is 30 m (50 m with RS422 adapter).

Only the 880 operator panels with 12" RGB analog monitors (RMOS operator panels) can be used as 2nd and 3rd operator panels.

An external 24 V power supply must be connected to the interface keyboard module of the 2nd and 3rd operator panels. This voltage must be present before switching on the 1st operator panel.

Cable break

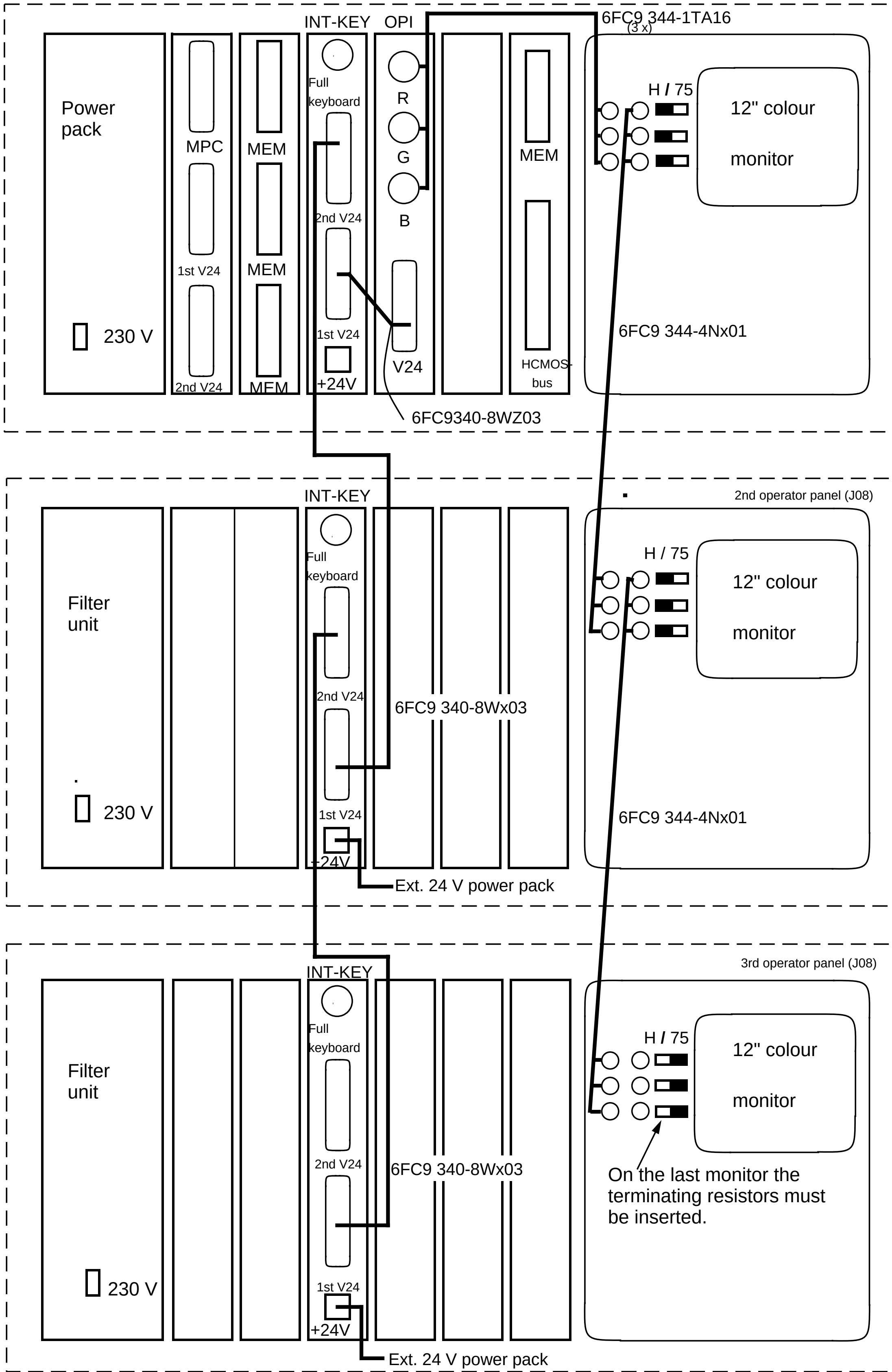
A cable break is identified by the handshake signals of the 1st RS232C (V.24) interface being recognized as inactive by the firmware.
This fault is indicated by the 4 operator panel LEDs lighting up consecutively (in other words the LEDs on the 1st operator panel are activated in the case of a cable break between the OPI and keyboard control module, those on the 2nd operator panel light up if the handshake signals between the 1st and 2nd operator panels are missing, and those on the 3rd operator panel are activated if the signals between the 2nd and 3nd operator panels are missing). The control does not have to be powered up again once the cable break has been eliminated.

Machine control panel

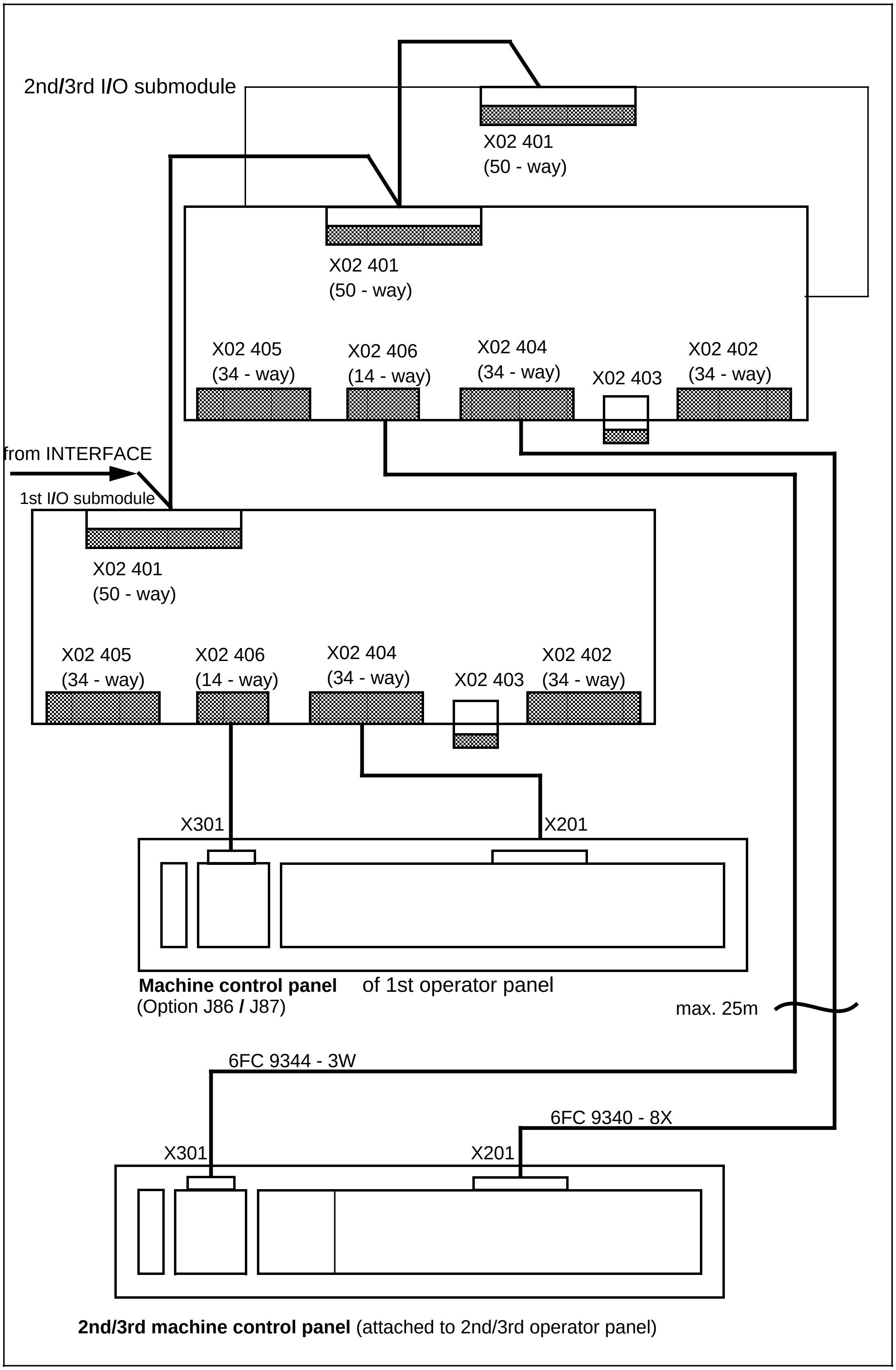
Machine control panels also have to be attached to the 2nd/3rd operator panel to allow the axes to be traversed from the various operator panels as well. These machine control panels are connected to the interface of the 1st operator panel by means of I/O submodules. Appropriate provisions must then be made in the PLC program so that only the signals of the machine control panel that is attached to the active operator panel are transferred to the interface.

Refer to Interface Description Part 2, Connection Conditions, for the rack assignment of the operator panels and the necessary cables.

Interface of 2nd/3rd operator panel



Interface of 2nd/3rd machine control panel



11.15 Tapping without compensating chuck

The TAPPING WITHOUT COMPENSATING CHUCK function is an Option.

11.15.1 Corresponding data

NC MD 274	Cycle number for G84/L84
NC MD 931	Activate differential measurement
NC MD 1260*	Feedforward control factor for tapping without compensating chuck
NC MD 1320*	Servo gain (K_V) factor for tapping without compensating chuck
NC MD 1324*	Time constant dynamic feedforward control factor for tapping without compensating chuck
NC MD 521* bit 4	C axis mode cancelled after M02/M30
DB10-25 DW13 bit 4	TAPPING WITHOUT COMPENSATING CHUCK signal
SD 40-43	Setting data for differential measurement (parameterization)

"Tapping without compensating chuck" Option

"Spindle package" Option (contained in Basic Version with T version)

11.15.2 Function description

The objective of this function is to do without the expensive compensating chucks that are often used when tapping.

It also offers the following benefits:

- Higher speeds can be achieved without a compensating chuck
- The final drilling depth can be complied with exactly
- Larger working area without compensating chuck

The basic concept behind this function envisages the spindle being switched to the rotary axis mode (e.g. C axis) for thread tapping before interpolating with the infeed axis. For this reason a C axis has to be assigned to the spindle (see Function Manual "Extended Spindle Functions").

The interpolation is executed by the new tapping cycle L841.

With thread tapping the following rotary axis machine data are replaced by a different set of parameters if the function was selected:

- Servo gain (K_V) factor (NC MD 252* 1320*)
- Feedforward control (NC MD 312* 1260*)
- Time constant for dynamic feedforward control (NC MD 392* 1324*)

If you use this second set of parameters, you can adapt the rotary axis to the requirements of tapping separately.

11.15.3 Programming and function sequence

The switchover of the spindle to the C axis mode is initiated with G203 and G204 (modal). Deselection is with G205 (reset state).

- G203 Selection of C axis mode, clockwise rotation
- G204 Selection of C axis mode, counter-clockwise rotation
- G205 Change from C axis mode to spindle mode M05

Programming

The following syntax must be observed:

G203**/**204 <Rotary axis (angle input)> <Infeed axis> S<Value>

e. g.: G203 C180 Z S150

No further G functions, preparatory functions or auxiliary functions must be programmed in this block. The angle input for the rotary axis can be omitted: 0 degrees applies in this case. The infeed axis is programmed without a value.

When switching automatically to the C axis mode, the spindle must already be in the required gear stage.

In conjunction with the switchover, the rotary axis approaches a defined angular position as programmed.

If G203**/**204 is active, the subroutine stored under NC MD 274 (standard L841) is automatically called when L84**/**G84 is programmed.

The cycle is initialized with R parameters R2 to R10 (same meaning as with L84).

The switchover from the rotary axis to the spindle mode is initiated with G205. Since only one spindle can be in the rotary axis mode per channel, the rotary axis does not have to be stated when deselecting.

G205 initiates the following:

- Change from C axis mode to spindle mode
- Switch back to first set of parameters
- PLC signal TAPPING WITHOUT COMPENSATING CHUCK

In the event of RESET/M02/M30, the rotary axis mode is cancelled (G205 is generated) if NC MD 521* bit 2 = 1, otherwise it is retained and G203/204 remains active, as does the 2nd set of parameters of the infeed axis.

Since the rotary and infeed axes interpolate with thread tapping, the effective servo gain (K_V) factors of the two axes must be the same. Dynamic feedforward control can also be used to reduce the following error.

Note:

When installing, the mixed I/O module 6FX 1138-4BA01 can be used to output the following errors on a pen recorder (refer to Section 8 of the Function Manual "Electronic Gearbox" for information on the analog outputs).

11.15.4 Notes on settings for tapping without compensating chuck

When working with the "tapping without compensating chuck" function, the tapping (linear) axis and the rotary axis that belongs to the spindle have to matched to one another. A parameterizable service cell has been introduced for this purpose:

- Parameterization under "setting data" with "analog output" softkey

- The settings are made with setting data SD 40, SD 41, SD 42 and SD 43
  SD 40: Global axis number of rotary axis belonging to spindle
  SD 41: Global axis number of tapping (linear) axis
  SD 42: Numerator of thread pitch
  SD 43: Denominator of thread pitch.

  Example:

  Rotary axis is global axis No. 4 (C)
  Linear axis is global axis No. 3 (Z)
  Thread pitch is 1 mm/revolution = 1 mm/360°

  This results in the following settings:

  SD 40 = 4, SD 41 = 3,
  SD 42 = 1, SD 43 = 360.

MD 931 must contain the value 2 so that differential measurement is activated.

The difference in position is displayed, that is the thread error in the service display of the axis stated under SD 41 (tapping axis) instead of the contour deviation. The resolution is IS (MD 5002).

It is practical to make the setting by initializing a cycle with longer traversing blocks in the C and Z axes, e.g.

```
N05 G01 Fxxxx C36000 Z100 LF          = G 1mm / 360°
N10 G01 Fxxxx C0 Z0 LF
N15 @100 K-5 LF
N20 M30 LF
```

The difference cell is set to 0 by writing any setting data or pressing NC STOP**/**START with the axis stationary (e.g. override 0 %).

The difference cell can be balanced to approximately 0 by slightly modifying the servo gain (K_V) factor or multgain (preferably of the rotary axis).

If the difference cell runs up quickly owing to a special machine configuration, the thread pitch can be entered as a negative value (negative numerator or denominator).

The service cell can also be used generally for dynamically balancing 2 interpolating axes. Use with the "tapping **with** compensating chuck" function is also advantageous. In this case, the spindle number can also be entered in SD 40 in the following format:

SD 40 = 10x
- x → Global spindle number
- 1 → Spindle identifier

Program a normal tapping program for test purposes. The thread path must be long enough, however, to provide sufficient time for observing the offset (e.g. a few cm). The difference in position can be deleted during the first thread block with NC STOP**/**START.

With the second thread block (≙ removing the tap), the compensating chuck withdrawal can be read off directly in the service cell.

Note the following points when working with the service cell:

- The service cell display is correct only if setting data are modified before or in the traversing block (e.g. M0)   reset positional error.
- After optimization, MD 931 must be reset to 0 otherwise additional NC CPU processing time will be used up.

11.16 Distance-coded reference point markers

11.16.1 Corresponding data

NC MD 240*	Reference point value, reference point for leadscrew error compensation
NC MD 284*	Reference point cutoff speed
NC MD 396*	Absolute encoder offset
NC MD 564* bit 0	Reference point approach direction (only necessary for G74)
NC MD 1300*	Basic distance for coded reference point markers
NC MD 1304*	External pulse multiplication
NC MD 1808* bit 2	Encoder absolute system opposite to machine system
NC MD 1808* bit 3	Absolute offset valid
NC MD 1808* bit 4	Measuring system 1 distance-coded
NC MD 1808* bit 5	Measuring system 2 distance-coded
DB32 DW(k) bit 12	REFERENCE POINT REACHED signal

11.16.2 Function description

Linear scales with distance-coded reference point markers have a reference point track as well as an incremental track. The distances between two reference markers in each case are defined differently. The absolute position of the axis can be determined by passing over two reference marks and counting the number of increments that were passed over.
Two adjacent reference markers can be passed over in any direction. A reference point cam is no longer needed.

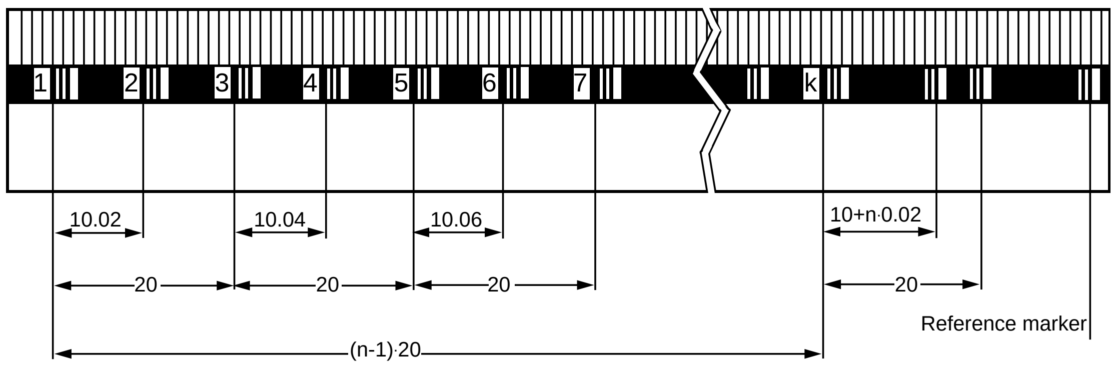
The "distance-coded reference point markers" function does not affect the reference point approach for the encoders used thus far.

The following measuring circuits must be used for processing distance-coded reference point markers:

- SPC measuring circuit, order No. 570 214 92xx.01
- HMS measuring circuit, order No. 570 456 900xx.xx

Example:

Linear scale with coded reference point markers



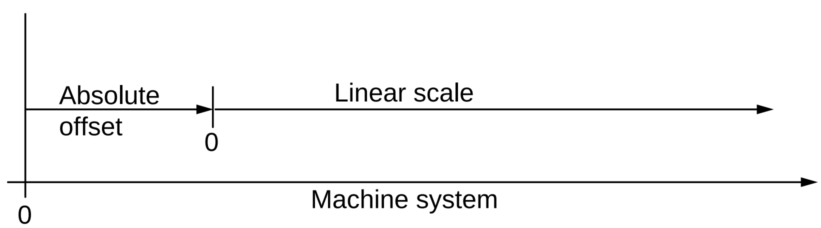
The procedure for reference point approach with distance-coded reference markers is as follows:

1) Selection of desired mode for reference point approach
"REFPOINT" mode followed by actuation of any direction key or execution of the G74 command in the "AUTOMATIC" and "MDI AUTOMATIC" modes.

2) The selected axis traverses in the designated direction at cutoff speed (NC MD 284*) (or several axes at the same time).
The axis decelerates at the 2nd reference mark.

3) The absolute position in the (linear) scale system can be calculated with the aid of the directional information, i.e. the direction in which the linear scale was traversed.

4) The axis position in the machine system is calculated depending on the attachment of the scale (same or opposite direction to machine system) and the entered absolute offset.



Note:

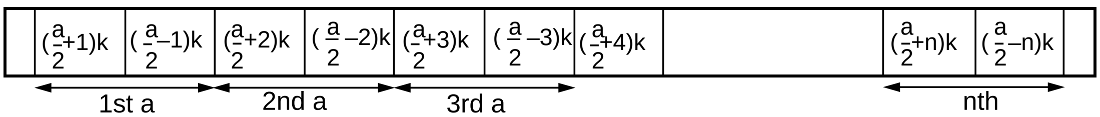
If the axis is already located between the penultimate and final reference markers when the reference point approach is selected, the referencing procedure cannot be executed because 2 reference markers always have to be overshot.

11.16.3 Function start-up

11.16.3.1 Determining cutoff speed

The RPC block (measuring circuit) can process the counter readings for the 1st and 2nd reference markers properly only if the reference marker signals are separated by at least one servo cycle (NC MD 160 - 163). If the cutoff speed is so high that both reference markers are overrun in the same servo cycle, the evaluation is incorrect.

For this reason the cutoff speed of the axis (NC MD 284*) must be reduced so that the smallest reference marker distance is traversed in a time that is greater than one servo cycle.



a: Basic distance
k: Division period

The smallest possible distance on the scale between two adjacent reference markers is calculated as follows:

$$x_{min} = \frac{a}{2} \cdot k - \frac{\text{Measuring length of scale in m}}{a}$$

a: Basic distance [in multiples of the division period]
k: Division period [in µ]

The maximum possible speed for the reference point approach is calculated as follows:

$$V_{max} \quad \frac{X_{min}}{T_{LR}}$$

X_{min}: Smallest possible reference marker distance
T_{LR}: Max. position control cycle

The value for the maximum possible speed v_{max} for the reference point approach is entered in machine data MD 284* (cutoff speed).

Example:

Calculation of maximum possible speed for a **Heidenhain linear scale LS704C**

Measuring length: 3040 mm
Division period k: 20 µm = 0.02 mm
Basic distance a: 1000 1000*k = 20 mm
Max. servo cycle T_{LR}: 8 ms

$$x_{min} = \frac{a}{2} \cdot k - \frac{\text{Measuring length of scale}}{a} = \frac{1000}{2} \cdot 0.02\text{ mm} - \frac{3040\text{ mm}}{1000} = 6.96\text{ mm}$$

$$V_{max} = \frac{X_{min}}{T_{LR}} = \frac{6.96\text{ mm}}{8\text{ ms}} = 52.2\ \frac{\text{m}}{\text{min}}$$

Entry for the **cutoff speed**: MD 284* (axis) 52 m/min.

Machine data MD 296* (reference point approach speed) is not evaluated.

11.16.3.2 Attachment of (linear) scale

It must be ascertained how the linear scale is attached in relation to the machine system. The relevant identifier is entered in MD 1808*.

NC MD 1808*. bit 2 = 0 Linear scale same direction as machine system
NC MD 1808*. bit 2 = 1 Linear scale opposite direction to machine system

11.16.3.3 Determining absolute offset

The absolute offset is derived from the displacement between the machine zero and the 1st reference point marker on the linear scale. With the axis in any position the absolute offset is the difference between the position measured in the machine reference system (e.g. with laser interferometer) and the current position in the display.

The absolute offset can be calculated as follows:

- Reset NC MD 396* of absolute offset to zero.
- Reference point approach (manual in REFPOINT mode). Brake the axis to zero speed after passing over two reference markers.
  This obtains the absolute position in the reference system of the linear scale (display position).
- Determine current axis position relative to the machine system zero with a laser interferometer.
- Calculate absolute offset:

	Measured or known current	axis position
Absolute offset =	Actual value display	in scale system
	in machine system	

-: Scale and machine system in same direction
+: Scale and machine system in opposite directions

- Enter calculated absolute offset in NC MD 396*.
- Set NC MD 1808* bit (absolute offset valid) to "1". If this bit is not set, the NC-PLC signal "REFERENCE POINT REACHED" is not set on conclusion of the reference point approach.

11.16.3.4 Inputs for various machine data

- NC MD 1300* — The basic distance between the coded reference point markers on the linear scale must be entered in this machine data.
- NC MD 1808* bits 4,5 — It must be stated here whether the 2nd measuring system that is in use or available is equipped with distance-coded reference markers on the linear scales.
- NC MD 240* — This machine data normally contains the position of the reference point. This is not necessary when using distance-coded reference markers. The value entered here is now relevant only as the reference point for leadscrew error compensation, in which no compensation is allowed.

11.17 Setpoint/actual value assignment with SINUMERIK 880 GA2

Corresponding data

NC MD 200*	1st measuring system actual value assignment
NC MD 220*	1st measuring system backlash compensation
NC MD 364*	1st measuring system pulses variable incremental weighting
NC MD 368*	1st measuring system traversing path variable incremental weighting
NC MD 1116*	1st measuring system HMS pulse multiplication
NC MD 1388*	2nd measuring system actual value assignment
NC MD 1228*	2nd measuring system backlash compensation
NC MD 1208*	2nd measuring system pulses variable incremental weighting
NC MD 1212*	2nd measuring system traversing path variable incremental weighting
NC MD 1204*	2nd measuring system HMS pulse multiplication
NC MD 1216*	2nd measuring system switchover tolerance
NC MD 384*	Axis setpoint assignment
NC MD 400*	Spindle actual value assignment
NC MD 460*	Spindle setpoint assignment
NC MD 1824* bit 1	2nd measuring system available

Function description

The SINUMERIK 880 GA2 allows the use of two measuring systems (actual value inputs) and two setpoint outputs for one axis or spindle/C axis combination. These actual value inputs and setpoint outputs can be freely located on several measuring circuits. A check for multiple assignments is executed only within a SERVO CPU. The system is unable to identify multiple assignments that exist across several SERVO CPUs.

The user must observe the following restrictions:

Associated setpoint/actual value inputs (e.g. actual value input 1, setpoint output 2) of an SPC 214 measuring circuit can be assigned different axes/spindles only if these axes/spindles are defined on the same SERVO CPU.

The common 1st measuring system (NC MD 200*) is the reference system with both axes and spindles with C axis. This reference system determines the accuracy of the position control. The parameterization of the C axis pulse weighting is thus effective in the spindle mode as well.

The following measuring system assignment applies for the **axes**:

Axis:	1st measuring system	NC MD 200*
	2nd measuring system	NC MD 1388*
	Setpoint output	NC MD 384*

- The 1st measuring system is always assigned to the axis as standard.
- If you wish to use a 2nd measuring system for an axis, it must be enabled with NC MD 1824* bit 1.
- In the case of axes, the 2nd measuring system can be selected with the "2nd measuring system" interface signal (DB 32, DW_{k+2}). A switchover tolerance threshold (NC MD 1216*) is monitored. In the event of deviations between the two measuring systems that exceed the tolerance threshold, the switchover request is refused and alarm 1016* (MS switchover not possible) is issued. Each of the two systems is assigned its own backlash compensation value. The variable incremental weighting is also entered separately for each measuring system.

- The reference point approach is executed with the currently selected measuring system.
- When using measuring systems with distance-coded reference markers, no more than one measuring system can be thus equipped.

The following measuring system assignment applies for the **spindle** and **spindle/C axis combination**:

C axis mode:	1st measuring system	NC MD 200*
	1st setpoint output	NC MD 384*
Spindle mode:	2nd measuring system	NC MD 400*
	2nd setpoint output	NC MD 460*

- With spindle mode (controlled, M19), the 2nd measuring system and 2nd setpoint output are always active.
- With C axis mode, the 1st measuring system and 1st setpoint output are active as standard. The 2nd measuring system (spindle encoder) can be selected with the "2nd measuring system" interface signal (DB 32, DW_{k+2}) in the C axis mode as well. The switchover tolerance (NC MD 1216*) is ineffective in this case. It can be advantageous to switch to the 2nd measuring system (spindle encoder) with thread tapping cycles as synchronization with the C axis encoder is not then necessary.
- Note that the machine data that belong to the 1st measuring system are also active when the 1st measuring system is active.
- When using a double-track encoder for the spindle and C axis mode, the zero markers of the two encoder systems can be offset. When installing, this offset has to be compensated by means of a zero mark offset (NC MD 459*). Also note that both measuring systems have the same feedback polarity (sign).
- If only one measuring system is used for the spindle and C axis (NC MD 200* = NC MD 400*), only the cutoff frequency of the spindle encoder (NC MD 462*) is active, not that of the C axis (NC MD 308*). Different pulse weightings, on the other hand, remain active.

With spindles/C axes, also note that the 2000 units offset that generally applies between the reference point and the zero mark during reference point approach is ineffective. If an axis is redefined as a C axis, therefore, the reference point is displaced by 2000 units (MS). This must be compensated with the reference point offset (NC MD 244*).

The following applies to backlash compensation and leadscrew error compensation: If the spindle and C axis encoders are different, the backlash and leadscrew error compensation are active only in the C axis mode. If the encoders have the same definition, the backlash and leadscrew error compensation are active in both the spindle and the C axis mode.

- Only one setpoint output is generally defined for a spindle/C axis combination (NC MD 384* = 460*).
- Dies ist unabhängig davon, ob ein oder zwei Geber verwendet werden.With spindle/C axis combinations the zero mark offsets NC MD 244* and 459* must be identical so that the position in the M19 mode is the same as the position in the C axis mode (M19S0 $\widehat{=}$ C0). This applies irrespective of whether one or two encoders are being used.

11.18 SIPOS absolute encoder

11.18.1 Functions

The SIPOS encoder system consists of a multiturn absolute encoder which functions absolutely when switched on and incrementally during operation.

To reduce the dimensions of the encoder, part of the encoder electronics are located on a submodule, the absolute submodule. The absolute submodule is plugged on the HMS measuring circuit module.

After each POWER ON, the control requests the encoder absolute value from the absolute encoder and then calculates the absolute position of the machine. The absolute encoder then functions in exactly the same way as a standard incremental encoder with amplified sine and cosine output signals.

The coded disc information (cyclic absolute) and the value from the battery backed tachometer are taken to determine the absolute information. The buffer battery is located on the absolute submodule and stores the information for three axes in each case.

Encoder key data:

Max. sampling frequency:	500 kHz
Increments on disc:	2500
Revolution information::	16 bits
Accuracy on POWER ON:	1 µm at a 10 mm spindle pitch (10000 incr./rev)
Error information:	8 bits
Battery back-up:	5800 hours for 3 encoders, built in battery monitoring

Even during operation, the back-up battery level is checked every 10 minutes and a message displayed on the control if a fault occurs.

If an additional back-up battery is installed in the encoder, the measuring circuit module and the encoder cable can be replaced without loss of absolute information. This battery back-up lasts for up to 5 hours.

11.18.2 Hardware requirements

The absolute encoder submodule is plugged onto the HMS measuring circuit in the same way as conventional servo command modules.

The absolute encoder submodule is equipped with its own processor which determines the values for 3 axes in each case.

The absolute encoder back-up battery which is also for 3 axes, is located on the absolute submodule.

The SIPOS absolute encoder can only be used in conjunction with the HMS measuring circuit module (high-resolution measuring system).

11.18.3 Synchronizing the absolute encoder with the machine absolute system

After installation or after replacing the absolute encoder, the measuring system must be synchronized with the machine system in the same way as with any incremental system. The absolute encoder is set up with the following NC machine data.

- **Axial machine data**

  NC MD 240* reference point value
  NC MD 396* absolute offset

- **Machine data bits:**

NC MD 1808*	bit 0 =	1	axis with SIPOS absolute encoder POWER ON required after setting this bit
	bit 2 =	0	absolute encoder same direction as the machine system
		1	absolute encoder opposite direction to machine system
	bit 3 =	0	absolute offset not valid
	bit 3 =	1	absolute offset valid

Definition of terms:

Same direction: The absolute encoder value rises and falls with the machine absolute value.

Opposite direction: The absolute encoder value rises as the machine absolute value falls and vice versa.

Direct measuring system: The absolute encoder is a position measuring system and is connected to the leadscrew, for example.

Indirect measuring system: The absolute encoder is used to measure position and speed and is therefore connected to the motor.

There are two synchronization methods; these are explained in the following subsections.

11.18.3.1 Synchronization in "Reference point approach" mode

Like any standard encoder, the SIPOS absolute encoder has A and B tracks and a zero pulse.

Similar to the incremental system, the encoder is synchronized with the machine system by combining the zero mark of the encoder with the reference point cam signal.

A distinction can be made between **two cases** by setting NC MD 1808* bit 0 = 1 ("Axis with SIPOS absolute encoder") during reference approach.

Note:

POWER ON is necessary after setting NC MD 1808* bit 0 = 1 (axis with SIPOS absolute encoder) because the absolute encoder value is requested and the machine position is calculated only if this routine is executed.

Case 1: NC MD 1808* bit 3 = 0

If the bit "Absolute offset valid" is not set, reference point approach is executed as for an axis without absolute encoder. With "Reference point reached", the calculated absolute offset is transferred to the axial NC MD 396* and NC MD 1808* bit 3 "absolute offset valid" is automatically set for the relevant axis.

The absolute offset is calculated according to the following equation:

Machine system = SIPOS absolute encoder value + absolute offset

Absolute offset = machine system - SIPOS absolute encoder value

Where:

- Machine system = desired absolute position = reference point value
- SIPOS absolute encoder value = displayed absolute position (actual value)

Note:

If NC MD 396* has a value other than 0, this value must be taken into account because it is contained in the SIPOS absolute encoder system (= displayed actual value).

Suggestion: Set NC MD 396* before synchronization.

Case 2: NC MD 1808* bit 3 = 1

If bit "Absolute offset valid" is set, reference point approach is suppressed. The condition prevails until the user resets the bit.

In the case of the function G74 "Reference point approach from part program", referencing is not carried out for the second case and processing resumes with the following block.

Note:

Reference point approach can be implemented again once NC MD 1808* bit 3 has been deleted.

11.18.3.2 Setting up without a reference point

Where there is no reference cam, e.g. for reasons of cost.

NC MD 1808* bit 0 = 1 is set, i.e. a SIPOS absolute encoder is installed.

The machine is traversed manually in "JOG" or "incremental" mode to the customer's chosen reference point.

The reference point value is entered in NC MD 240*.

Calculating the absolute encoder offset:

Machine system = SIPOS absolute encoder value + absolute offset

Absolute offset = machine system - SIPOS absolute encoder value

Where the following is valid:

- Machine system = desired absolute position = reference point value
- SIPOS absolute encoder value = displayed absolute position (actual value)

Notes:

- If NC MD 396* has a value other than 0, this value must be taken into account because it is contained in the SIPOS absolute encoder system (= displayed actual value).

  Suggestion: Set NC MD 396* before synchronization.
- When the SIPOS absolute encoder is set up with reference point approach an overrun control is carried out automatically. An overrun results because the absolute encoder is absolute to $\pm 2^{15}$ revolutions and it cannot be ruled out that on setup an overrun of 0 to 2^{15} revolutions from the zero position will occur.
- When setting up without a reference point the customer must carry out the overrun control. All absolute values used must then have the format ± 99999999 input units.
- If a value larger or smaller than ± 99999999 input units results from the calculation of the absolute offset by the machine system - SIPOS absolute encoder value, it must be corrected as follows:

  Absolute offset > 99999999: Absolute offset - 199999999
  Absolute offset <-99999999: Absolute offset +199999999

Example:

a) Calculated absolute offset = 130000000
correction: absolute offset = 130000000 - 199999999 = - 69999999

b) Calculated absolute offset = - 130000000
correction: absolute offset = - 130000000 + 199999999 = +69999999

11.18.4 What happens on warm restart (POWER ON)

When NC MD 1808* bit 0 "Axis with absolute encoder" is set, NC MD 1808* bit 3 "Absolute offset valid" is checked for the relevant axes. If both bits are set, the axis-specific interface signal "Reference point reached" is set on warm restart.

11.18.5 Special case "Parking axis"

The axis-specific interface signal "Parking axis" deletes the signal "Reference point reached" even for an axis with a SIPOS absolute encoder. If NC MD 1808* bit 3 is set, referencing is again suppressed. Not until a renewed POWER ON is the absolute value again transferred.

Functions that require a referenced system, such as leadscrew error compensation and software limit switches, are no longer effective.

11.18.6 Absolute encoder errors

The absolute encoder errors are displayed axis-specifically in the control with the error message "**Absolute encoder defective**".

The exact type of error is displayed in the "Service number" line in the Service status display menu. The error number is displayed.

Error numbers are explained in the section "Absolute encoder errors".

Notes:

- When the absolute encoder is **first switched on**, various absolute encoder numbers are displayed for a period of approx. 10 minutes. This is caused by the back-up battery of the absolute encoder which only establishes error-free communication with the control after a certain charge time.
- All absolute encoder errors are transmitted from the absolute encoder to the control on start-up. It is not possible to acknowledge the absolute encoder errors (e.g. using the reset key). The control can only be informed that the error has been eliminated after a renewed POWER ON.

11.18.7 Comments

- NC MD 1808* bit 3:
  - Software limit switches and leadscrew error compensation are enabled.
  - The leadscrew error between the reference point and the absolute position are included in the calculation.
- Reference point approach with a SIPOS absolute encoder is only possible when NC MD 1808* bit 3 has been reset.
- Information as to whether the SIPOS absolute encoder is in the same or opposite direction as the machine system is required by the control for calculating the absolute value without reference point approach. Once the SIPOS absolute encoder has been synchronized it is very easy to determine the direction of the SIPOS absolute encoder. This is done by traversing the axis concerned to any desired position and starting up the control.

  If, after setting up, positive approach and POWER ON, a larger absolute encoder value is displayed, the system is in the same direction and does not have to be corrected. If a smaller absolute encoder value is displayed, NC MD 1808* bit 2 = 1 must be set. The correct position is then displayed after the next POWER ON.
- If the mechanical traverse direction is changed with NC MD 564*, NC MD 1808* bit 2 must also be adapted correspondingly.

11.18.8 Absolute encoder errors

With SIPOS, errors can occur in the encoder as well as in the absolute submodule located on the HMS measuring circuit module.

Please note that a faulty connection between the encoder and the absolute submodule can trigger nearly all the error messages because it is along this connection that communication takes place. This transmission line should therefore always be checked first when an error occurs.

Section 10.4 contains a list of the error numbers.

Note:

When sending in faulty components for repair, always state the error No.

12 Alarms

12.1 List of alarms

This Section contains all of the alarms that displayed in the control.

12.2 Alarm numbers and groups / clearing of alarms

NC alarms

- POWER ON alarms
- V.24 (RS232) alarms
- RESET alarms/axis-specific
- RESET alarms/general
- RESET alarms/spindle-specific
- ERASE alarms

Note:

With POWER ON-RESET (switch on control) all NC alarms are cleared.

PLC alarms

- PLC error messages
- PLC operational messages

Note:

The PLC-MD must be used to determine whether alarms 6000 to 9999 are displayed as error or operational messages (or not at all).

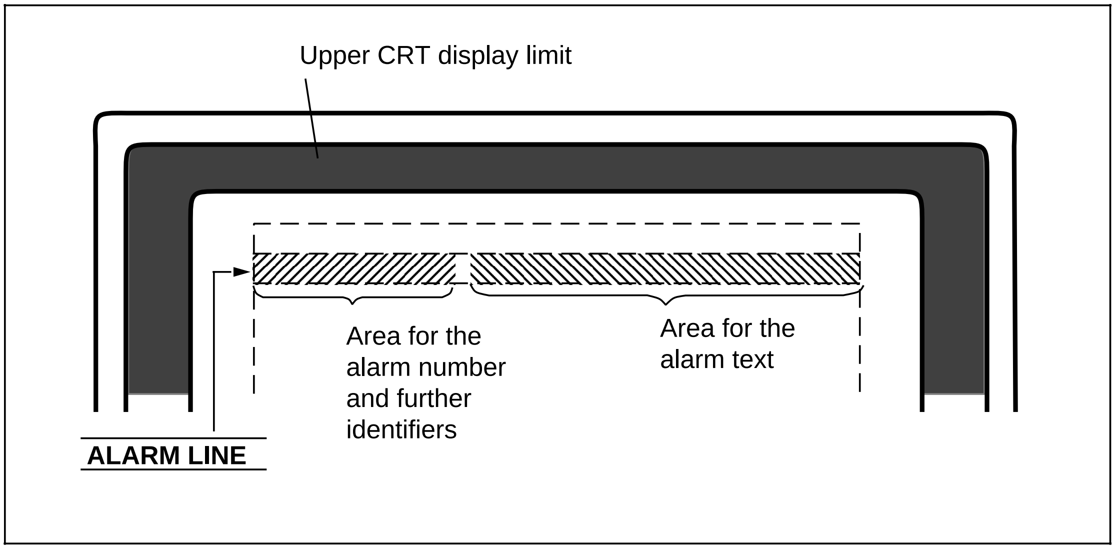
Cycle alarms

- SIEMENS cycle alarms
- User cycle alarms

12.2.1 Display of alarms on the CRT

Messages from the monitoring system are displayed in the alarm line of the CRT display.

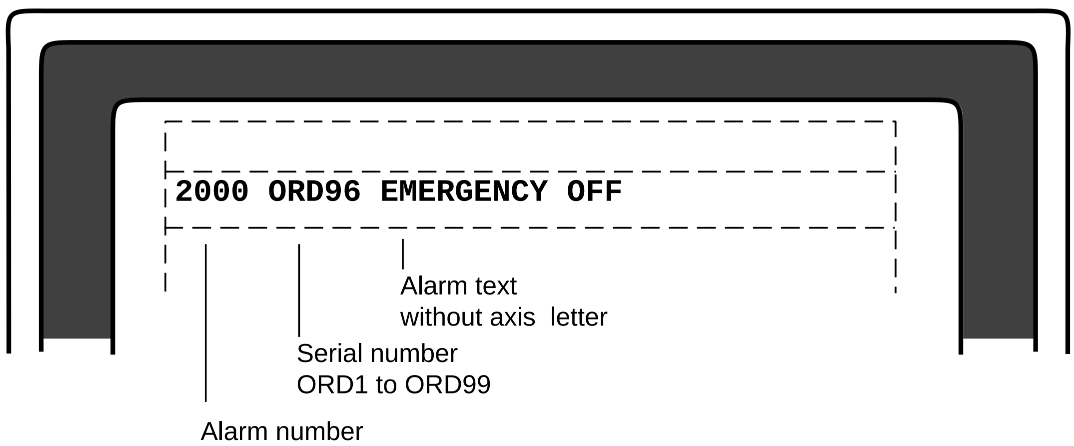
The alarm line is the second display line from the top.

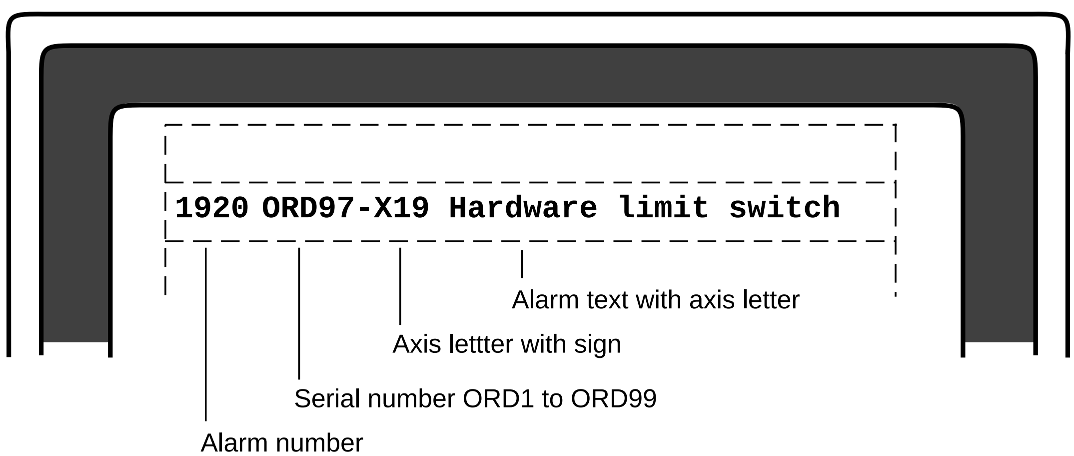


12.2.2 Display representation

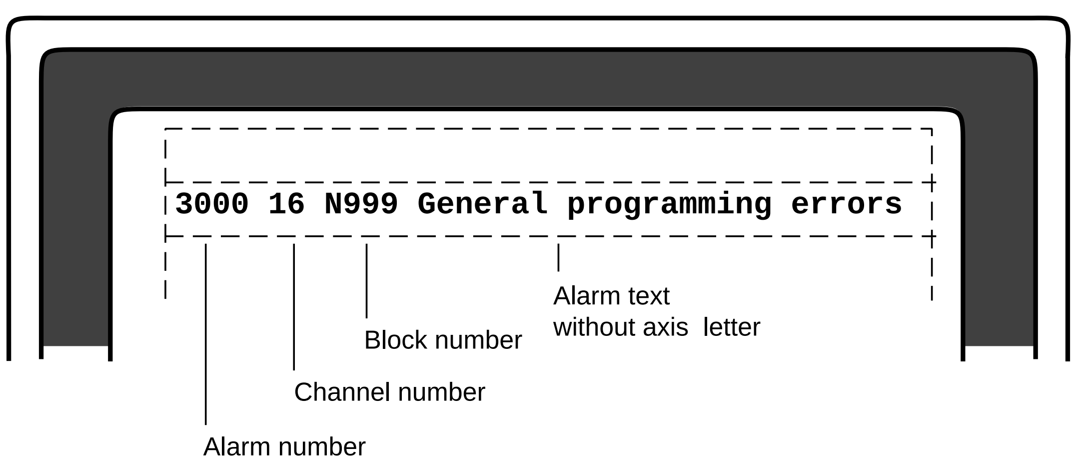
There are 3 types of display representation for alarms:

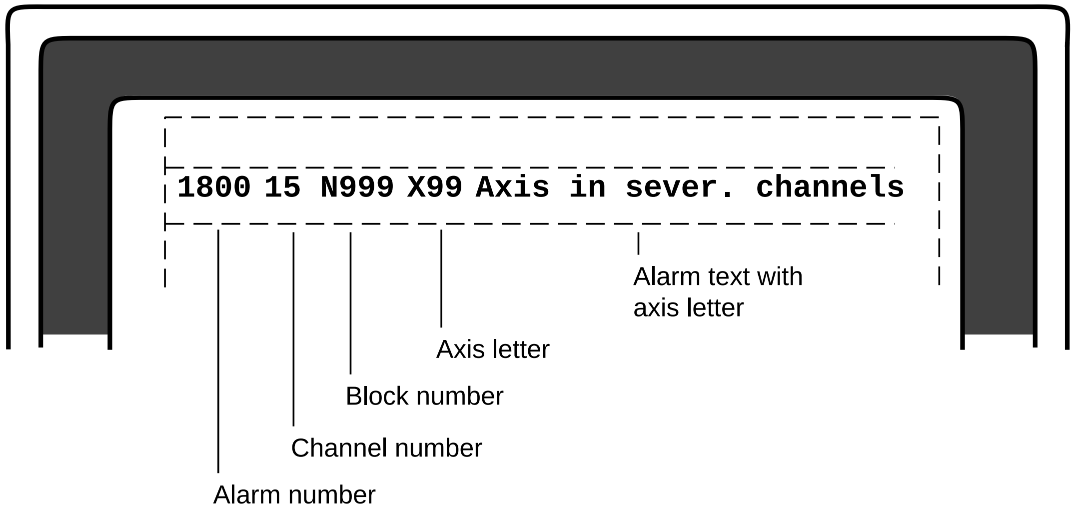
1. Serial number-referred alarms





2. Block related alarms





Note:

Axis- and spindle-specific alarms in this Operator's Guide are marked with an asterisk in the last decimal place. Example: Alarm number 156*

The asterisk stands for the relevant axis or spindle. The following assignments apply:

Spindles	**Axes**
1 = 1st spindle	0 = 1st axis
2 = 2nd spindle	1 = 2nd axis
3 = 3rd spindle	2 = 3rd axis
4 = 4th spindle	3 = 4th axis
5 = 5th spindle	4 = 5th axis
6 = 6th spindle	.
	.
	.
	23 = 24th axis

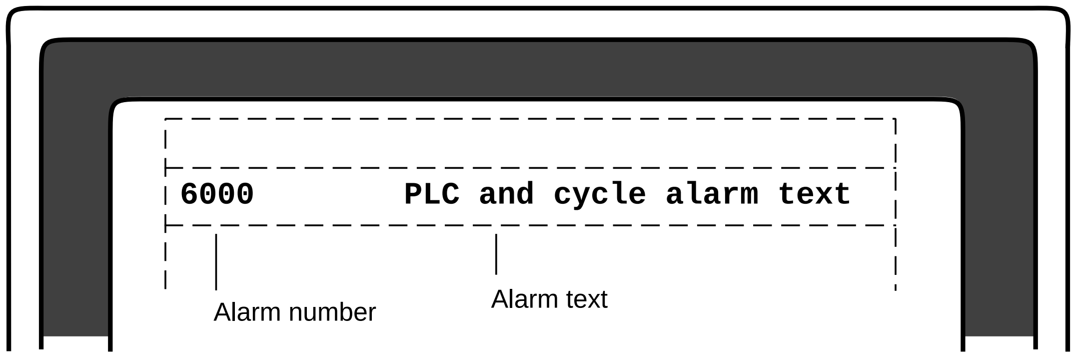
Examples for spindles: Alarm 225* spindle speed too high

2251	Alarm refers to the 1st spindle
2252	Alarm refers to the 2nd spindle
2256	Alarm refers to the 6th spindle

Examples for axes: Alarm 156* setpoint speed too high

1560	Alarm refers to the 1st axis
1565	Alarm refers to the 6th axis
1570	Alarm refers to the 11th axis
1583	Alarm refers to the 24th axis

3. PLC and cycle alarms



Note:

PLC and cycle alarms are determined by the machine tool manufacturer or the user himself. They are not included in the alarm description therefore.

12.2.3 Alarm numbers/cancellation of alarms

Alarm number	Kind of alarm	Alarm is cancelled by
1	General alarms	Pressing the "Acknowledgement" key
2 to 15	General alarms	POWER ON
16 to 36	V.24-Alarme	Selecting 1. Menu call for data IN-OUT 2. "Data" softkey 3. "Stop" softkey
40 to 95	General alarms	POWER ON
1000 to 1211 1240 to 1251	Axis-specific alarms	Pressing the "Reset" key
1280 to 1371	Axis-specific alarms	POWER ON
1440 to 1971	Axis-specific alarms	Pressing the "Reset" key
2000 to 2195	General alarms	Pressing the "Reset" key
2250 to 2263	Spindle-specific alarms	Pressing the "Reset" key
2270 to 2273	Spindle-specific alarms	POWER ON
2280 to 2427	Spindle-specific alarms	Pressing the "Reset" key
3000 to 3246	General alarms	Pressing the "Acknowledgement" key
4000 to 4299 5000 to 5299	Cycle alrarms	Pressing the "Acknowledgement" key
6000 to 9999	PLC error messages or PLC operational messages	Pressing the "Acknowledgement" key
10000 to 12031	Axis-specific alarms	POWER ON or pressing the "Reset" key
20000 to 20309	Spindle-specific alarms	POWER ON or pressing the "Reset" key

POWER ON means switcheing the control off and on again.
Please note the information supplied by the machine tool manufacturer.

12.2.4 Diagnostics/selection of further alarms

When the diagnostics react, the reason could be **several different faults occurring at the same time**.

Only the alarm with the **lowest alarm number** is displayed in the alarm line.

If you need an overview of any other existing alarms/messsages, proceed as follows:

In any of the 7 operating modes, first select the **DIAGNOSTICS** softkey function and then **ALARM MESSAGES**

Select softkey function	**NC ALARM**
or	**PLC ALARM**
or	**CYCLE ALARM**
or	**PLC MESSAGE**

12.2.5 Effects of alarms

In the section ”List of alarms/alarm description” the four types of effect are listed. The detailed meaning of the effects is explained below.

- **Interruption of processing**

  The program is interrupted and can be resumed at the interrupt point after the fault has been remedied.

  With channel-specific alarms, processing is interrupted only in the channel concerned.

- **NC START inhibit**

  NC START is not enabled when an alarm is active.

  The current processing is not stopped. However, if processing is interrupted (e. g. by NC STOP or Single block) it cannot be restarted until the cause of a fault has been remedied and the alarm cleared.

  With channel-specific alarms, NC START is inhibited only for the channel concerned.

- **Cancellation of Mode Group Ready**

  The MODE GROUP READY signal is cancelled for the master channel. As a result, maximum braking current is applied to the axes or spindles of the mode group for rapid deceleration (Rapid stop).

- **Opening the NC ready relay**

  The output on the dual/multiport is disabled. The ready state of the control can only be reestablished via POWER ON.

12.2.6 Listing of alarms/alarm description

The following list is based on the presumption that servicing is not carried out by the operator.

The following footnotes apply in this Section:

c) Valid for 880 GA1 up to and including SW6

d) Valid from 880 GA2, SW1

1	**Battery alarm-power supply**
Scan:	• On POWER ON • Cyclic
Effect:	• None
Explanation:	• The battery voltage has dropped so far that backup (of the program memory, for example) cannot be guaranteed if the control is switched off
Remedy:	Replace the battery next to the power pack under power and acknowledge the alarm with the ”Acknowledge key” after making spot checks in the part program memory for variable data
Caution:	The control must not be switched off until the battery has been replaced, otherwise stored data will be lost

2	**Overtemperature**
Scan:	Cyclic
Effect:	Output of temperature error at PLC user interface flag 24.2
Explanation:	• The fan exhaust temperature is higher than 55 °C (permissible range 0 °C to 55 °C)
Remedy:	Check temperature in the cabinet

4	**Incorrect unit system**
Scan:	• On POWER ON • After modification of NC MD
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• An illegal combination of machine data MD 18000 (display resolution) and MD 502 (input resolution) has been selected. Both these data must contain the same units system. • The alarm is displayed with reference to serial numbers.
Remedy:	Check combinations of machine data and correct them. Then clear the alarm with POWER ON.

5	**Incorrect MIB parameter**
Scan:	On formatting the user program memory; UM FORMAT softkey
Effect:	Only the MIB parameters 0 to 199 are defined
Explanation:	• The alarm is displayed with a serial number. • The R and MIB parameters take up so much space in the user memory that the user program memory has become smaller than 16 Kbytes • MD 4 < MD 5 overlapping of both MIB areas • MD 4 < 200 standard MIB area exceeded • MD 5 > 500 MIB buffer area exceeded
Remedy:	Correct NC MD 4 or NC MD 5 and reformat the user program memory

6[c)]	**Battery alarm machine data**
Scan:	• On POWER ON • Cyclic
Effect:	None
Explanation:	• The alarm is displayed with a serial number. • The battery voltage has dropped so far that backup of the NC MDs cannot be guaranteed
Remedy:	Replace the NC MD card with a new one and re-enter the NC MDs and PLC MDs (MD buffered by control battery from start of 1991)
Caution:	The control must not be switched off before the replacement has been made, otherwise stored data will be lost

c) *Up to and including SW 6*

7	**Battery alarm operator panel**
Scan:	• On POWER ON • Cyclic
Effect:	None
Explanation:	• The battery voltage has dropped so far that backup of the setup data for the Terminal Emulator cannot be guaranteed
Remedy:	Replace battery in the operator panel under power and acknowledge the alarm with the CANCEL key after the setup data have been checked and corrected if necessary
Caution:	The control must not be switched off until the battery has been replaced, otherwise stored data will be lost

8	**Wrong axis/spindle assignment**
Scan:	• After modification of NC MD • On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	NC machine data for axis assignment (MD 200*) and/or spindle assignment (MD 400*) inorrectly input or implemented. Error in MD 461* C axis definition.
Remedy:	Check and correct machine data for axis assignment and spindle assignment. Cancel alarm with POWER ON.

9	**Memory too small for UMS**
Scan:	On POWER ON
Effect:	Illegal UMS
Explanation:	• Excessive modification of the standard area,available RAM area insufficient
Remedy:	Avoid excessive modification of the standard area

10	**UMS error**
Scan:	On POWER ON
Effect:	NC START disabled
Explanation:	• The alarm is displayed with a serial number • UMS selected but no UMS inserted or EPROMS on UMS board are empty • UMS RAM not yet loaded
Remedy:	• Insert correct UMS • Check NC MD 5015 bit 6

11	**Wrong UMS identifier**
Scan:	On POWER ON
Effect:	NC START disabled
Explanation:	• The alarm is displayed with a serial number • UMS RAM not yet loaded • Wrong EPROM inserted • Linking error (operating system)
Remedy:	• Load UMS RAM • Insert correct EPROM • Relink UMS

12	**Part program memory wrongly formatted**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number. • A fault has been ascertained on the memory module or a hardware repair has been made and the part program memory not yet reformatted
Remedy:	Reformat memory ("CLEAR PART PROGRAM" softkey) or replace memory

13	**Check NC MD**
Scan:	On POWER ON
Effect:	• The alarm is displayed with a serial number • Opening of NC Ready Relay
Explanation:	Disparities revealed by plausibility checks on runup
Remedy:	Check MD in the field "block number". See notes with machine data in question.

14	**Check PLC MD**
Scan:	On POWER ON
Effect:	• The alarm is displayed with a serial number • Opening of NC Ready Relay
Explanation:	Disparities revealed by plausibility checks on runup
Remedy:	Check MD in the field "block number". See notes with machine data in question.

16	**Parity error RS 232 C (V.24)**	Soft-key
Scan:	On reading data into the RS 232 C (V.24) interface	
Effect:	• RS232 C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• Alarm is displayed with the block number. • The alarm can only arise if the setting data (SD) ”with parity bit” applies. The character commenced (8 data and 1 parity bit) has the wrong parity. This alarm has nothing to do with character parity errors RS232 C (V.24) in ISO or EIA punched tapes (Alarm 23)	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

17	Overflow error RS 232 C (V.24)	Soft-key
Scan:	On reading data into the RS 232 C (V.24) interface	
Effect:	• RS 232 C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The external device has sent a new character although the NC has not yet processed the old one	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

18	Frame error RS 232 C (V.24)	Soft-key
Scan:	On reading data into the RS 232 C (V.24) interface	
Effect:	• RS 232 C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• Alarm is displayed with the block number • Number of stop bits is incorrect • Baud rate is incorrect	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

19	I/O device not ready RS 232 C (V.24)	Soft-key
Scan:	On request of data from the NC via the RS 232 C (V.24) interface	
Effect:	No files are read in	
Explanation:	• Alarm is displayed with the block number • The DSR signal from the external device is Low	
Remedy:	• Start external device • Do not use DSR	

22	**Time watch-dog RS 232 C (V.24)**	Soft-key
Scan:	Following an NC request to output or read in data; after a delay of 60 seconds	
Effect:	No data are received or output	
Explanation:	• Alarm is displayed with the block number • The NC cannot output a signal for 60 seconds – External device blocks the "CTS" signal (**c**lear **t**o **s**end) for longer than 60 seconds – External device does not send DC1 within 60 seconds when control signals (DC1 to DC4) are used • The NC has not received a signal for 60 seconds	
Remedy:	• Check external device and switch on • Check cable and plug in • Switch off time watch-dog (SD bit)	

23	**Character parity error RS 232 C (V.24)**	Soft-key
Scan:	On reading in data via the RS 232 C (V.24) interface of the NC	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • Depending on the definition of the start of program "%" or "EOB", the NC automatically determines the ISO or EIA code on receipt of this character and thus the character parity On checking the following characters, it has been established that one character did not have the set parity	
Remedy:	Check punched tape and/or format output of connected data device. The number of the block containing the error is displayed.	

24	**Invalid EIA character RS 232 C (V.24)**	Soft-key
Scan:	On reading in data via the RS 232 C (V.24) interface of the NC	
Effect:	• RS232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • An EIA character with the correct parity was read in, but the character was **not** defined in EIA code	
Remedy:	• Check punched tape • Check SD (EIA code for "@") • Check SD (EIA code for ":") • Check SD (EIA code for "=") • Check SD (EIA code for "]") • Check SD (EIA code for "[") • Check SD (EIA code for ",")	

26	**Block > 120 characters RS 232 C (V.24)**	Soft-key
Scan:	On reading in data via the RS 232 C (V.24) interface of the NC	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The part program block that has been read in contains more than 120 characters. Only the actually stored characters are counted (no spaces, no CR, ...)	
Remedy:	Divide block into 2 or more blocks	

27	**Data input disabled RS 232 C (V.24)**	Soft-key
Scan:	On reading in data via the RS 232 C (V.24) interface of the NC	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with the block number • An attempt has been made to read in NC MDs in normal mode • An attempt has been made to transfer UMS data to the NC although the UMS has not been enabled or not inserted • An attempt has been made to read in a subroutine which already exists as a cycle in the EPROM • "Cycle inhibit" interface signal present	
Remedy:	• Read in NC MDs in installation mode or enter password • Insert UMS module and fit with UMS submodules or enable UMS via machine data • Reset "cycle inhibit" interface signal (DB 48)	

28	Circ. buffer overflow RS 232 C (V.24)	Soft-key
Scan:	On reading in data into the NC via the RS 232 C (V.24) interface	
Effect:	• RS 232 C (V.24) transfer interrupted • The last blocks declared invalid	
Explanation:	• Alarm is displayed with the block number • Data transfer speed so high that more characters are transferred than can be processed by the NC • RTS signal has no effect on the input device (RTS causes the input device to STOP) • Transmission speed (baud rate) too high	
Remedy:	• Use RS 232 C (V.24) transmission instead of 20 mA • Use 3rd serial interface	

29	Block > 254 charac. RS 232 C (V.24)	Soft-key
Scan:	On reading in tool data into the NC via the RS 232 C (V.24) interface	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The block that has been read in has more than 254 characters. All characters read in are counted (including blanks, CR, LF ...)	
Remedy:	Divide block into 2 or more blocks	

30	Part program memory overflow RS 232 C (V.24)	Soft-key
Scan:	On reading in programs via the RS 232 C (V.24) interface of the NC	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The maximum part program memory space is full	
Remedy:	Clear old programs to provide memory space for reading in new programs. The block number containing an error is displayed.	

31	**Program memory full RS 232 C (V.24)**	Soft-key
Scan:	On reading in programs via the RS 232 C (V.24) interface of the NC	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with the block number • The available part programm memory is full	
Remedy:	• Read out and clear old part programs that are not needed anymore to provide memory space for new part programs • Use larger part program memory if necessary	

32	**Data format error RS 232 C (V.24)**	Soft-key
Scan:	On reading in data via the RS 232 C (V.24) interface of the NC	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The permissible number of decades after an address is not correct • The decimal point occurs in the wrong place • The part programs or subroutines are not defined or concluded correctly (check heading!)	
Remedy:	Check the program to be read in (the number of the block containing the error is displayed)	

33	**Different programs same number RS 232 C (V.24)**	Soft-key
Scan:	On reading in programs via the RS 232 C (V.24) interface of the NC	
Effect:	No data have been entered/stored	
Explanation:	• Alarm is displayed with the block number • If a new program is read in which has the same program number as the one already stored in the NC, both programs are compared and they are different, the NC outputs Alarm 33. The new program is **not stored.**	
Remedy:	• Delete old program (DELETE) • Rename old program (RENAME)	

34	**Operator error RS 232 C (V.24)**	Soft-key
Scan:	On starting another RS 232 C (V.24) interface when the other one is already active	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with the block number • The RS 232 C (V.24) transmission is started in the NC and the PLC sends DATA START once more to the NC	
Remedy:	Restart RS 232 C (V.24) transmission	

35	**Reader error RS 232 C (V.24)**	Soft-key
Scan:	Only if setting data are set for the Siemens reader	
Effect:	• RS 232 C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The Siemens reader has output a group error message	
Remedy:	• Restart RS 232 C (V.24) data transfer • If the error recurs: replace Siemens reader	

40	**COM CPU fault**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number. • Hardware or software fault in the COM CPU area • No PLC-CPU fault reported • Incorrect or fault EPROM submodule inserted	
Remedy:	Replace hardware and/or software	

41	NC CPU fault
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Hardware or software fault in the range of the respective NC CPU • Incorrect or faulty EPROM submodule inserted
Remedy:	• On installation: check jumper on the bus board • Replace hardware and/or software

42	SERVO CPU fault
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Hardware or software fault in the range of the respective SERVO CPU • Processor overloaded • Measuring circuit module fault • Incorrect or faulty EPROM submodule inserted
Remedy:	• Check NC MD 156 or 160 to 163 • Reduce the number of axes/spindles on the SERVO CPU • Deselect tool nose radius compensation or assign axis to a different SERVO CPU • Replace hardware and/or software

43	**PLC-CPU fault**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Hardware or software error in the PLC or NC COM area • PLC in STOP status • NC does not assume Reset status and continues displaying the status of the instant of interruption • PLC machine data error • Error in the PLC user program • Incorrect or faulty EPROM submodule inserted
Remedy:	• Read out I-STACK • Establish cause of error using the error list in the installation lists • Replace hardware and/or software

44	**Part program memory not available**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • Part program memory missing or not recognised or does not reach the minimum size
Remedy:	• Check specified memory size (machine data) • Clear memory and format

46	**Incorrect TO parameter display**
Scan:	Cyclic
Effect:	none
Explanation:	• The alarm is displayed with a serial number • NC MD 13 too large
Remedy:	Modify NC MD

47	**`Illegal TO assignment`**
Scan:	Cyclic
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Value of NC MD 210 is greater than 16 • TO starting numbers in NC MDs 211 to 226 have not been entered in ascending order • Input value in NC MDs 1040 to 1047 is greater than the number of TO ranges or 0 is defaulted in the TO range
Remedy:	Check NC MDs

57d)	**`SERVO timeout`**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • Set position control time for axes or spindle too low
Remedy:	Resynchronize (switch off/on), modify position control time

58d)	**`Parameterization error`**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Input setpoint 0 and follow-up mode
Explanation:	Dual assignment of measuring circuit (see alarms 1000* and 2016*)
Remedy:	Correct machine data MD 200* and/or MD 400*

d) From GA2, SW 1

59[d)]	**SIMODRIVE failure**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Input setpoint 0 and follow-up mode
Explanation:	• Cable to processor module pulled out during operation or processor module switched off
Remedy:	Resynchronize in cyclic mode (switch off/on)

60[d)]	**Drive PM without voltage**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Input setpoint 0 and follow-up mode
Explanation:	SINUMERIK switched on before processor module
Remedy:	Alarm automatically cancelled when processor module switched on

61[d)]	**Drive not synchronized**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Input setpoint 0 and follow-up mode
Explanation:	Negative acknowledgement for processor module triggered by hardware

d) From GA2, SW 1

63 Connection fault

Scan: Cyclic

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready
- Opening of NC Ready Relay

Explanation:
- The alarm is displayed with a serial number
- The transmission connection between the COM CPU and the operator panel is interrupted. Normal operation is no longer possible.

Remedy:
- Check the fiber-optic or copper cable between the operator panel and the central controller
- Check COM or operator panel CPU

64 Incorrect checksum

Scan: On POWER-ON

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready
- Opening of NC Ready Relay

Explanation:
- Alarm is displayed with the block number
- An EPROM error has been found on switching on the control:
  - Checksum of one card is wrong. The card number is displayed as block number. This number corresponds to the number of the Servo CPU with the faulty EPROM. Block number 0 corresponds to the NC CPU (error LED flashes).
  - Checksum of Restart EPROM wrong

Remedy:
- Check card slots and correct if necessary
- Check EPROMS

65[c)] Wrong slot

Scan: On POWER ON

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready
- Opening of NC Ready Relay

Explanation:
- Alarm is displayed with the block number
- On switching on the control it has been found that one module has been inserted in the wrong bus slot

Remedy: Check module slots and replace modules if necessary

c) Up to and including SW 6; from GA2, SW 1 the entire system software is stored on a single memory card.

66[c)]	**Wrong software system**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • On switching on the control it has been found that the wrong software system has been inserted in the control e. g. 810 software system instead of 880 software system
Remedy:	Insert correct software system

67	**1st computer link not ready**
Scan:	Cyclic or after POWER ON
Effect:	Message frame interchange between the host computer and the NC is not possible.
Explanation:	• The alarm is displayed with a serial number • Host computer and NC are not synchronous caused by a wrong specification or a fault on the interface module. Message frame interchange is not possible.
Remedy:	• Check configuration of interface module • Check machine data specified for the computer link • Check whether host computer is ready or has been connected

68	**2nd computer link not ready**
Scan:	Cyclic or after POWER ON
Effect:	Message frame interchange between the host computer and the NC is not possible.
Explanation:	• The alarm is displayed with a serial number • Host computer and NC are not synchronous caused by a wrong specification or a fault on the interface module. Message frame interchange is not possible.
Remedy:	• Check configuration of interface module • Check machine data specified for the computer link • Check whether host computer is ready or has been connected

c) Up to and including SW 6; from GA2, SW 1 the entire system software is stored on a single memory card.

69	Servo CPU not available
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • The servo circuit of an axis/spindle has been defined in MD 200* or MD 400* on a non-existing servo CPU.
Remedy:	Correct MD 200* or MD 400*

70	Fault in at least one channel
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • A wrong assignment in machine data has been chosen on installation. The NC is not operational without channel assignment. At least 1 channel must be assigned to each NC CPU.
Remedy:	Check channel assignment machine data and correct (MD 100*)

71	NC CPU not available
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. One channel has been assigned to a non-existing NC CPU.
Remedy:	Check channel assignment machine data and correct (MD 102*)

72	**One channel too many in mode group**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. Too many channels have been assigned to one mode group.
Remedy:	Check mode group assignment machine data and correct (MD 100*)

74	**One channel too many in the NC CPU**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. Too many channels have been assigned to one NC CPU.
Remedy:	Check channel assignment machine data and correct (MD 102*)

75	**Too many control circuits on the CPU**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Too many real axes and spindles have been defined in the machine data.
Remedy:	Check machine data and correct

76	**Wrong measuring circuit assignment**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. One axis/spindle has been assigned to a wrong measuring circuit or a wrong measuring circuit module.
Remedy:	Check machine data and correct

77	**Illegal mode group no. of axis**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Alarm generated only if mode group No. = 0 or > 8
Remedy:	Check machine data 360* and correct

78	**Illegal mode group no. of spindle**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Spindle has been assigned to a wrong mode group
Remedy:	Check machine data 453* and correct

79	**Illegal mode group no. of channel**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Channel has been assigned to a wrong mode group
Remedy:	Check machine data 100* and correct

80	**Illegal NC no. of channel** c)
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • Channel has been assigned to a wrong NC CPU
Remedy:	Check machine data 102* and correct

81	**More than two 2D interpolations**
Scan:	On each block change
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • The NC is an export version. In the NC, 2 axes (2D interpolation) have been programmed in more than 2 channels in one program block.
Remedy:	• Modify program • Do not run more than 2 programs simultaneously

c) Up to and including SW 6

82	**Wrong simulation channel selected**
Scan:	On pressing the UM FORMAT softkey
Effect:	None
Explanation	• The alarm is displayed with a serial number • Graphics simulation is not possible because a wrong value has been specified in NC MD 173
Remedy:	Correct NC MD 173 (input value 1 or 2)

83	**Wrong no. of SWD parameters**
Scan:	On pressing the UM FORMAT softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Graphics simulation is not possible because a wrong value has been specified in NC MD 172
Remedy:	Correct NC MD 172 (only input value 8 permissible)

84	**Incorrect coupled axis grouping**
Scan:	• On POWER ON • On warm restart
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • An incorrect coupled motion grouping has been specified in NC MD 876 to 899 or 5156 to 5182, e. g. – Axes do not belong to the same mode group – Axes have different position control resolutions – Different axis type (linear axis/rotary axis) – Axes are not available (NC MD 564* bit 7=0) – Axes are fictitious axes – Leading axis=following axis
Remedy:	• Correct NC MDs 876 to 899 • Correct NC MDs 5156 to 5182

85	Wrong coupled motion combination
Scan:	• On POWER ON • On warm restart
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • The undefinable bit combination “01” has been entered in the NC MDs 5156 to 5182
Remedy:	Correct NC MDs 5156 to 5182

87	Illegal software limit switch
Scan:	On modification of MD
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • An excessive value has been entered in NC MDs 224*to 236*. The maximum traversing range of the individual axes is determined by the axis-specific position control resolution set and the input resolution.
Remedy:	Correct NC MD 224* to 236* if necessary

88	Interpolation greater than 3D[c] 4D[d]
Scan:	On processing part programs in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • More than 4 axes are programmed in one block in the NC part program block or the 5D function is not provided
Remedy:	• Check part program • Divide program block into two separate blocks • Have function option checked by a service expert and retrofit if necessary

c) Up to and including SW 6
d) From GA2, SW 1

89 More than two 3D interpolations

Scan: On processing part program blocks in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready
- Opening of NC Ready Relay

Explanation:
- Alarm is displayed with the block number
- In the NC, more than 3 axes are programmed in one program block in more than 2 channels

Remedy:
- Modify program
- Process up to 2 programs simulaneously

100* Illegal leadscrew error compensation grid spacing

Scan: Cyclic

Effect:
- NC START disabled
- Cancellation of Mode Group Ready

Explanation:
- The alarm is displayed with a serial number
- The permissble range (1 to 1000 lead screw error compensation points) has been exceeded in axis traversing movements.
- Lead screw error compensation for rotary axes. A wrong value for the relevant axis has been entered in NC machine data 324*.

  Formula: 360/input value = number of grid spacings
  The number of grid points must be an interger!

Examples:	NC MD 324*=10	360/10=360000 grid points
	NC MD 324*=12	360/12=300000 grid points
	NC MD 324*=14	360/14=257143 grid points

  The value 14 is not permissible because the result must always be an interger

Remedy: Modify NC MD 324*

104* DAC limit[c)] Set speed warning limit[d)]

Scan: Cyclic

Effect:
- NC START disabled
- The drive will oscillate if too high a velocity is entered
- Alarm 156* ”Setpoint speed too high” is triggered if there is a fault in the drive

Explanation:
- The alarm is displayed with a serial number
- The DAC setpoint is higher than entered in NC MD 268* (maximum DAC setpoint). It is not possible to increase the DAC setpoint further.

Remedy:
- Traverse more slowly
- Check actual value (encoder)
- Check NC MD 268*
- Check the drive actuator

---

c) Up to and including SW 6
d) From GA2, SW 1

108*c)	**Overflow with variable increment weighting**	
Scan:	On each axis movement	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is cancelled after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with a serial number • If a value is entered in NC MDs 364*, 368* (pulse weighting) which is not 1:1, the actual part value must be multiplied by the control. With rapid traverse of the axis, the register will overflow in the case of a fault. The reference point is lost.	
Remedy:	• Reduce maximum velocity (NC MD 280*); maximum velocity depends on NC MDs 364* and 368* • Check MDs 364* and 368* • Check actual values (encoder)	

c) *Up to and including SW 6*

112*	Zero speed control
Scan:	• On standstill • On clamping • On deceleration
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is cancelled after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • The following error could not be cleared faster than the time entered in NC MD 156 • On clamping, the limit defined in NC MD 212* was exceeded • A mechanically clamped axis has been pushed out of position • Fault in the control device (actuator), at the tacho-generator, at the motor, in the NC measuring circuit hardware or at/on the encoder • Wrong position feedback polarity direction on installation
Remedy:	• NC MD 212* (clamping tolerance) must be greater than NC MD 204* (exact stop limit) • NC MD 156 (servo enable cutoff delay) must be large enough to allow the following error to be cleared within this period (this only applies if NC MD 372* = 0) • NC MD 372* (zero-speed control delay) must be large enough to allow the following error of the axis to be cleared within the period entered • Check actual values (encoder) and position control polarity • Reduce acceleration (accelaration/delay so large that the circuit voltage in the drive will reach the permissible limit)

116*	Contour monitoring
Scan:	When machining in AUTOMATIC/MDA, but **not :** • When accelerating • When decelerating • At speeds lower than that in NC MD 336* (contour speed)
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • With a speed greater than that in NC MD 336* the tolerance band in NC MD 332* has been exceeded • When accelerating or decelerating the axis has not reached the new speed within the time defined by the servo gain factor
Remedy:	• Increase the tolerance band in NC MD 332* • Check the servo gain factor • Check the optimization of the speed controller • Check the actual values (encoder) • Check the free movement of the axes • Reduce acceleration

120*[c)] Illegal axis specification

Scan:
- On standstill
- On clamping
- When decelerating

Effect:
- Processing interruption
- NC START disabled
- Servo enable is cancelled after the time in NC MD 156 has elapsed (servo enable relay drops out)
- Follow-up mode

Explanation:
- The alarm is displayed with a serial number
- Missing setting via MD200* or MD384* of the axis concerned
  Example: MD2000=01020101 and MD3840=00000000
- The module number set in MD 200* or MD 384* is larger than the number of measuring circuit modules available
  Example: MD2000=04010000 and only 3 measuring-circuit modules inserted
- The connection number set in MD 200* or MD 384* is larger than the number of available connections on the module in question
  Example: MD3840=02070000, the 2nd measuring-circuit module is an SPC module and has only 6 terminals
- Connection number for an input is assigned to an output and vice versa
  Example: MD3840=01030000, the 1st measuring-circuit module is an HMS module on which terminal no. 3 is an input terminal
- Input and output assignment is not compatible with the insertion sub-module
  Example: MD2000=01040101, the 1st measuring-circuit module is an HMS module with output submodule servo command 6FX1132-5BAxx in its submodule slot 1

Remedy:
- Check and correct MD 200* and 384* of the axis concerned. These machine data must both be specified or be zero. They must also agree with the hardware configuration.

c) *Up to and including SW 6*

128*	Measuring circuit not available
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • In NC MD 200* or 400* measuring circuits have been assigned to axes or spindles which have been recognised by the NC as non-existent after POWER-ON RESET • Measuring circuit module removed or defective
Remedy:	• Check machine data • Check measuring circuit module

132*	Control loop hardware
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • The measuring circuit differential signals – are not in phase – have a short-circuit to earth – are missing • With distance-coded reference marks, the reference point approach was not executed within 2x "basic distance of reference marks"
Remedy:	• Check that the measuring circuit connector is inserted • By plugging in the measuring circuit short-circuit connector it can be checked whether the measuring circuit module is in working order • Check the differential signals with an oscilloscope • Replace the measuring sensors • Check NC MD 200*, 384* • Check IAR MD 100*c)
Caution:	The alarm can only be deleted with POWER ON

c) Up to and including SW 6

136*	**Measuring system dirty**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • In measuring systems with contamination signal (e.g. EXE), the measuring system signals a fault to the NC
Remedy:	Check the measuring system following the manufacturer's instructions
Caution:	The alarm can only be cancelled with POWER ON

140*	**Encoder monitoring**
Scan:	On axis motions (setpoint not equal to 0) if the function has been activated via machine data bit
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Only check 32 bit servo CPUs • Too many directional changes have been recognized in the actual position value between two position control timers. Possible causes are: – One encoder track is broken or connected to a fixed potential (typical fault) – The encoder is precisely located on an edge and the hardware tends to oscillate (effect of contamination) – Fault in the electronics leads to clattering of one track (effect of contamination)
Remedy:	Check encoder track

144*	**Zero mark monitoring**
Scan:	Cyclic, depending on specification of tolerance band for differential pulses
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • Pulses have been lost as a result of errors in transfer or other faults beyond the permissible tolerance band with reference to one encoder revolution. The tolerance band is checked by a reference meter.
Remedy:	• Check encoder pulses and establish permissible deviation • Check transmission path • Switch off monitoring temporarily by specifying a value greater than 16 in the machine data (NC MD 216*)

148* 152*	**Software overtravel (limit) switch plus** **Software overtravel (limit) switch minus**
Scan:	At each axis movement
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • The software limit switches are only active after reference point approach • Depending on the PLC interface signal 2ND SOFTWARE LIMIT SWITCH ACTIVE, the 1st or 2nd software limit switch has been approached • Traverse away from the limit switch in opposite direction
Remedy:	• Check NC MDs 224*, 228*, 232*, 236*

156*	**Set speed too high** c) **Set speed alarm limit** d)
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • A higher set speed has been output in the control than that specified in NC MD 264* • The motor cannot follow the speed command value • On installation: wrong position control direction • Excessive following error occurred because acceleration too great or owing to incorrect encoder adjustment (feedback polarity, weighting) • Malfunctions in driving unit • Mechanical clamping
Remedy:	• Check whether the value in NC MD 264* is higher than the value in NC MD 268* • Check the drive • Check the position control direction • Check the set speed cable • Check the actual values (pulse encoder) • Check IAR MD 124* and 136*

c) Up to and including SW 6
d) From GA2, SW 1

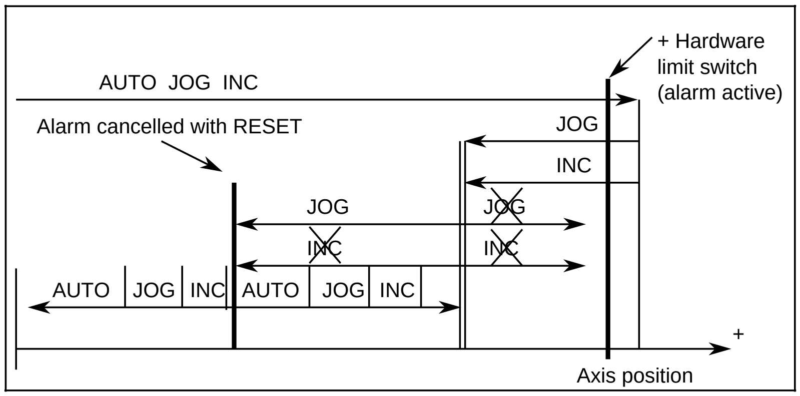
160*	Drift too high
Scan:	Only with semiautomatic drift compensation
Effect:	• NC START disabled • The green "Position not yet reached" LED lights up • No traverse movement possible
Explanation:	• The alarm is displayed with a serial number • The drift to be compensated by the NC itself has risen to over approx. 500 mV
Remedy:	• Execute drift compensation in NC MD 272* (see also Installation Guide) Operation: 1. Select NC MD 272* 2. Press the ”EDIT” key • Check that the drift has been correctly adjusted on the drive unit

164*	Synch. (coupled) axis programmed
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Despite the fact that coupled motion has been selected a following axis was programmed
Remedy:	Program program axis

168*	Servo enable refused for traversing axis
Scan:	At each axis movement
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • The axis-specific servo enable signal was removed by the PLC during traversing
Remedy:	Check the PLC program

172* **176***	**+ Working area limitation** **- Working area limitation**	
Scan:	Cyclic after reference point approach	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with a serial number • The working area limitation in the setting data has been reached	
Remedy:	• Check the working area limitation in setting data (selection via the "SETTING DATA" and "AXIAL DATA" softkeys) • Check G25**/**G26 in the part program and modify if necessary • Move relevant axis clear of limitation • Cancel ”Reference point reached” signal (e. g. define axis as ”parking axis”)	

180*	**Axis activated in several channels**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• Alarm is displayed with the block number • This alarm is displayed with reference to the channel and to the block • On simultaneous processing of two or more programs in different channels the same axis has been programmed in both programs (channels)	
Remedy:	• Check both programs • Insert L999 or @ 714 • Stop one channel with NC STOP	

188* 192*	**+ Hardware overtravel (limit) switch** **- Hardware overtravel (limit) switch**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Direction key disabled in approach direction
Explanation:	• The alarm is displayed with a serial number • Limit switch reached
Remedy:	• Traverse in opposite direction (correct operating mode)  • Check PLC user program • Check LIMITATION signals in DB 32

196*	**Synch. (coupled) axis assigned 2 x**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• NC START disabled • Processing interuption
Explanation:	• The alarm is displayed with a serial number • Two leading axes have been programmed whose following axes are identical Axis X axis Y Axis Z axis Y • Two leading axes have been programmed, one of these also being the following axis of the other leading axis Axis X axis Y Axis Y axis Z
Remedy:	Correct program

2000	Emergency Stop
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • The EMERGENCY STOP signal is output from the PLC to the NC
Remedy:	• Check with PLC STATUS • Check whether the "EMERGENCY STOP" cam was traversed to, or whether the "EMERGENCY STOP" switch was operated • Check PLC user program

2031	Weighting factor too large/small
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • On account of the weighting factor the actual axis speed is so high that the maximum permissible speed would be exceeded with the axis-specific position control resolution set
Remedy:	• Check NC MD 388* • Program lower speed • Reduce feedrate/rapid traverse override

2032 d)	Error in thread
Scan:	• On NC START and after block search • When switching to C axis mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the block number • Incorrect gear stage when switching to C axis mode • FOLLOW-UP MODE or PARKING AXIS present when selecting tapping for C axis • Selection not made from initial setting G205
Remedy:	• Program gear change • Check channel number • Check PLC signals, check NC MD 521*.4 • Cancel PARKING AXIS signal

d) From GA2, SW 1

2035	**Feedrate limitation**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The programmed speed is greater than the path speeds resulting from the maximum speeds of the axes • The programmed distance to go in the thread block is equal to zero
Remedy:	• Do not program a spindle speed with G95/G33 • Check NC MD 280* • Check part program

2036	**G35 thread lead decrease error**
Scan:	On thread cutting
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the block number • The pitch decrease in threading is so high that at the end of the thread, a diameter of 0 would arise
Remedy:	Program a lower pitch decrease or a shorter thread

2037	**Prog. S value too high**
Scan:	On processing part programs in AUTOMATIC or MDA mode
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The programmed spindle speed in AUTOMATIC is too high • An excessive feedrate has been calculated for thread cutting (G33/G34/G35/G37) as a result of the programmed spindle speed
Remedy:	Program a lower spindle speed

2038	Path feed too high
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• Alarm is displayed with the block number • The axis speed has caused the programmed path feed to increase so much that the maximum permissible axis speed would be exceeded with the position control resolution set
Remedy:	• Program a lower path feed • Check interpolation condition in the part program block

2039	Reference point not reached
Scan:	On starting a part program/program block in AUTOMATIC or MDA mode
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The reference point has not been traversed to by at least one axis and ”NC START” has been pressed in MDA or AUTOMATIC mode.
Remedy:	• Approach reference point • The alarm does not appear if NC MD 5004 "NC START without reference point" is set. • This alarm can be disabled axis-specifically in NC MD 560*

2041	Program not in memory
Scan:	On specification of a program number and subsequent NC START
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The pre-selected program is not in the memory • A non-existing subroutine is called in the main program • The contour for the stock removing cycle does not exist
Remedy:	• Look up directory • Check program

2042 Parity error in memory

Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • One or more characters in the memory have been corrupted and can no longer be identified. These characters are output in the "Correction block" or in the "EDITOR" as "?".
Remedy:	• Correct the program in "EDITOR" or delete the complete block and re-enter it if necessary

2043 Prog. error transformation

Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• NC START disabled • Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Programming of real axis with transformation selected • Programming of fictitious axes with transformation deselected • Selection of transformation although transformation has already been deselected • Programming of traversing movements in the selection block of transformation
Remedy:	Correct program

2044 External processing error

Scan:	On starting an external program
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The alarm is displayed: – if an external program selected is already being processed in another channel – if an interface (RS 232C (V.24) or File transfer) is to be started in several channels, or if the interface is already busy – if the 2nd interface of an interface pair (1st and 2nd RS 232C (V.24), 3rd and 4th RS 232C (V.24)) is to be started while the 1st interface is busy
Remedy:	• Check NC machine data 130* • Check the active interfaces

2045	**Program already available**
Scan:	On NC START, if "Processing an external program" has been selected
Effect:	Processing not started
Explanation:	• Alarm is displayed with the channel and block number • A program having the same program number as the external program to be processed is already in the part program memory
Remedy:	Rename the existing program or clear

2046	**Block > 120 characters**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • An "L_F" in the memory has been corrupted so that a block containing more than 120 characters exists
Remedy:	Insert "L_F" without deleting the complete block

2047	**Option not available**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • A function has been programmed that is not included in the function set of the control • Setting data 204* ("thread starting angle") has been edited although EXTENDED THREAD PACKAGE Option is missing
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary

2048	**Circle endpoint error**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The programmed circle end point does not lie on the circle • The end point is out by more than the value entered in NC MD 7
Remedy:	Correct program

2056	**Transf. centre travel**
Scan:	On processing in AUTOMATIC (Block search) or MDA mode
Effect:	• Processing interruption • NC START disabled • Block search interrupted
Explanation:	• Alarm is displayed with the channel and block number • A part program block has been programmed with selected transformation TRANSMIT that initiates a movement directly through the transformation centre • The path from the start to the end position of the block to be searched passes directly through the transformation centre
Remedy:	• Correct program • On block search: modify start position in JOG

2057	**Thread/revolutional feedrate option**
Scan:	On processing a part program block when this option is selected
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • A thread has been programmed using G33, G34, G35, G37 although this option is not available in the control • Revolutional feedrate G95 has been programmed • In the 880 T the "Revolutional feedrate" NC MD bit has not been set • A function has been programmed that is not included in the function set of the control
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary

2058	**3D interpolation option not available**
Scan:	On processing program blocks in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Three or more axes have been programmed in one block
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary

2059 G92 program error

Scan: On processing a relevant part program block

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- An illegal address character has been used
- The unit and working diameter factor is zero

Remedy:
- Check program block
- Check machine data

2060 TO, ZO program error

Scan: On processing program blocks in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- Tool type is 0 (no tool)
- A non-existent tool compensation number has been selected
- The values in the selected zero offsets or tool compensations are too large

Remedy:
- Check specifications in the program block
- Check machine data specifications
- Cancel alarm via the Reset key

2061 General program error

Scan: On processing a part program or program block in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- The input resolution determines the range limitation for the positional data. If the programmed path length exceeds the permissible path distance to go, an alarm is activated.

Remedy:
- Check the program section and correct the incorrect specification
- Check the input resolution selected

2062 Feed missing/not programmed

Scan: On processing program blocks in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- Revolutional feedrate G95 programmed greater than 50 mm/min
- No revolutional feedrate programmed
- No F value programmed
- No feedrate for soft approach/exit
- Zero is specified in MD 280* for the maximum speed of one axis

Remedy:
- Check specifications in program block
- Check machine data specifications
- Cancel alarm via the Reset key

2063 Thread lead (leadscrew) too high

Scan: On thread cutting with G33/G37

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- The thread lead can be specified in the program under I, J, K. The programmed value exceeds the permissible value depending on the specified display resolution.

Remedy:
- Correct program block whose block number and channel number is shown in the alarm display

2064 Rounding wrongly programmed for rotary axis

Scan: On processing in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled
- The programmed movement in the block is **not** executed

Explanation:
- Alarm is displayed with the channel and block number
- If you round to either half or full degrees on a rotary axis, the control will monitor whether the programmed positions correlate with the rounding

Remedy:
- Program the correct rotary axis position
- Check MD 560* bits 2 and 3
- Check whether the interface signal CLEAR DISTANCE TO GO has been set (no automatic rounding!)

Caution:
- In the JOG, INC modes, the control automatically rounds to valid values. In the AUTOMATIC or MDA modes, the control only monitors the programmed positions, without rounding itself.
- The INC mode is not permissible

2066 Thread lead increase or decrease too large

Scan: On processing in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- A thread lead or pitch increase or decrease of more than 16 mm/rev (0.6 inch/rev) has been programmed

Remedy: Program a smaller thread lead increase/decrease

2067 Max. axis speed = 0

Scan: On processing in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- The resulting maximum speed of one axis is equal to 0

Remedy: Check NC MD 280*

2069 5D TLC (tool length correction) not possible

Scan: On processing in AUTOMATIC mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- Cutter radius selected
- No linear interpolation selected
- Option or NC MD not set or set incorrectly
- ”3D interpolation” option not set
- Export version

Remedy:
- Check program
- Check machine data

2072 Incorrect input value

Scan: On processing in AUTOMATIC or MDA mode

Effect:
- Processing interruption
- NC START disabled

Explanation:
- Alarm is displayed with the channel and block number
- Programmed input for contour definition cannot be calculated

Remedy:
- Check program
- Check input values

2073	No intersection point
Scan:	On processing in AUTOMATIC mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Calculation of the contour definition gives no intersection point with the values programmed
Remedy:	Check programmed specifications

2074	Incorrect angle value
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Angle 360° is programmed • The angle value has no meaning for the contour described
Remedy:	Check programmed values and correct

2075	Incorrect radius value
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Radius value too large • Radius value not allowed for the contour described
Remedy:	Check programmed values

2076	Incorrect G02/G03
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Direction of circle not possible for the described contour
Remedy:	Correct program

2077	Incorrect block sequence
Scan:	On processing part programs in AUTOMATIC mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Several blocks are needed for calculation of contour definition – Block sequence not correct – Insufficient information (underdefined)
Remedy:	Check programmed values

2078	Incorrect input parameter
Scan:	On processing part programs in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Programmed parameter sequence not allowed • Parameter sequence not complete for the described contour Example: `N10... X60 B15` L_F (Z axis missing) `N20... X90 B10` L_F
Remedy:	Check program

2081	CRC/TNRC not allowed
Scan:	On processing in AUTOMATIC mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The following functions cannot be programmed with cutter radius/tool nose radius compensation selected: G33, G34, G35, G37, G58, G59, G92, G200, @714
Remedy:	• Preprogram G40 • Cancel with G41/G42 D00 (CRC/TNRC)

2082	**CRC plane not determinable**
Scan:	On processing a part program in AUTOMATIC mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The axes for the selected CRC plane do not exist • G41/G42 is selected and the plane definition (NC-MD 548*, 550*, 552*) or G16 programming is incorrect
Remedy:	• MD 548* • MD 550* • Check MD 552* • Select correct plane with G16

2087	**Coordinate rotation not allowed**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • On selection of coordinate rotation (G54 to G59) a circular movement (G02, G03) has been programmed in the next block • The settable zero offset (G54 to G57) has been changed after selection of coordinate rotation (angle 0 degrees) • The plane (G16, G17, G18, G19) has been changed after selection of coordinate rotation (angle 0 degrees)
Remedy:	• Correct program • The plane can only be changed if the angles of rotation and zero offsets programmed under G58 and G59 are zero • The settable zero offset and coordinate rotation (G54 to G57) can only be changed if the angles of rotation and zero offsets programmed under G58 and G59 are zero (Deselect if necessary)

2089	**SERVO 1 computing time**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary

2105	**SERVO 2 computing time**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary

2121	**SERVO 3 computing time**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary

2137	**SERVO 4 computing time**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary

2160	**Scale factor not allowed**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • The scale factor has exceeded the valid value range: – P is negative – P = 0 – P > 99.99999
Remedy:	Observe valid value range for scale factor: (P = 0.00001 to 99.99999)

2161	**Scale change not allowed**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • With a scale factor > 1 an axis position has been programmed which is so great that internal representation is no more possible
Remedy:	• Check programmed axis position • Reduce scale factor

2165	**Computing time exceeded**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with a serial number • The number of axes and spindles on the servo CPU is greater than the permissible maximum number. • The alarm is also set if the control recognizes the timing error, i. e. processing is no more performed in the IPO clock cycle. The set speeds are set to zero in the event of overrun of position control loops. All mode groups with axes on the relevant servo CPU are instantaneously stopped. The “Computing time exceeded” alarm indicates that the servo CPU is overloaded.
Remedy:	Report to Service

2171 Approach not possible

Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • No axis of the selected plane (G16, G17, G18, G19) has been programmed in the block following approach so that no vector can be calculated for tangential approach • @714 has been programmed in the selection block or in the subsequent block for soft approach
Remedy:	• Correct program (G147, G247, G347) by programming at least **one** axis of the selected plane in the block following approach

2172 Retraction not possible

Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • No axis of the selected plane (G16, G17, G18, G19) has been programmed in the block preceding retraction so that no vector for tangential retraction can be calculated • @714 has been programmed in the deselection block or in the block before • G48 programmed **without** preceding selection
Remedy:	• Correct program (G148, G248, G348, G48) by programming at least **one** axis of the selected plane in the block preceding retraction

2173 Wrong approach/retraction plane

Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • A change of plane is programmed in the block following selection (G16, G17, G18, G19) • A change of plane is programmed in the deselection block
Remedy:	• Correct program (change of plane)

2184	**`Illegal M function for C axis`**
Scan:	• During ramp-up • On warm restart • On MD changes
Effect:	The C axis drive for spindles on the 32 bit servo CPUs cannot be selected.
Explanation:	• The alarm is displayed with a serial number • The numbers of the M functions for selecting and deselecting C axis drive are determined via MD 260 and MD 261. These numbers must not coincide with the defaulted M functions
Remedy:	Valid values must be entered in MD 260 and MD 261

2189	**`Transformation undefined`**
Scan:	On selection of transformation
Effect:	• NC START disabled • Processing interruption
Explanation:	• Alarm is displayed with the block number • Type of transformation not defined • Transformation axes are in different mode groups • Option for selected transformation not available • Transformation selected in illegal channel • Transformation defined several times or wrongly defined • Transformation data block declared invalid by Alarm 3087 (error in transformation data)
Remedy:	• Check transformation data block • Check program • Check channel number • Have option checked by Service and retrofit if necessary

2190	**`Transformation axes assigned`**
Scan:	On selection of transformation
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• Alarm is displayed with the block number • A transformation has been selected whose real axes are also used in parallel-running transformation in another channel
Remedy:	• Wait for deselection of transformation in parallel-running channel • Check program

2191	Transformation at zero
Scan:	On selection of transformation in AUTOMATIC (Block search) or MDA mode
Effect:	• NC START disabled • Processing interruption • Block search is interrupted
Explanation:	• Alarm is displayed with the block number • Transformation has been selected at a moment when one or several of the axes involved in transformation were in the actual position ZERO. • On selection of TRANSMIT the X axis (transverse axis) must not have the actual position ZERO. This also applies to block search.
Remedy:	Set the real axes of the transformation to be selected to permissible actual positions prior to selecting transformation (on TRANSMIT the X axis to X 0)

2192	Axis is not a rotary axis
Scan:	In each G195 block
Effect:	• NC START disabled • Cancellation of Mode Group Ready
Explanation:	• Alarm is displayed with the block number • The axis name following G195 does not define a rotary axis
Remedy:	Program axis name of rotary axis together with G195

2194d)	FIFO not available
Scan:	• On POWER ON • On warm restart
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • FIFO channel assignment performed although FIFO Option not activated
Remedy:	• Set Option • Have option checked by Service and retrofit if necessary

d) *From GA2, SW 1*

2195d)	**Too many FIFO chan. defined**
Scan:	• On POWER ON • On warm restart
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • The FIFO memory can be assigned max. 2 channels per NC CPU
Remedy:	Check MD 558*

225*c)	**Spindle speed too high**
Scan:	Only with NC MD 520* set (encoder available)
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • The actual spindle speed is higher than that set in machine data or setting data
Remedy:	• Program a smaller S value • NC MDs 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 PLC gear stage • Check G92 S... at "v=constant" • Check spindle speed limitation setting data • Program G26 S ...

c) Up to and including SW 6
d) From GA2, SW 1

226*c) Control loop spindle - Hardware

Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Servo enable of spindle is removed after the time in MD 477* has elapsed (waiting time for servo enable)
Explanation:	• The alarm is displayed with a serial number • The measuring circle differential signals – are not in phase – have a short-circuit to earth – are entirely missing
Remedy:	• Check whether measuring circuit connector has been plugged • You can check by inserting the measuring circuit short-circuiting plug whether the measuring-circuit module is in operating order • Check the differential signals using an oscilloscope • Replace sensor

227*c) Spindle measuring system dirty

Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • On measuring systems with a contamination signal, the measuring system has signalled a fault to the NC
Remedy:	Check the measuring system

228*c) M 19 option not available

Scan:	On processing in AUTOMATIC or MDA mode or when processing an external program is specified
Effect:	Function is not executed
Explanation:	• The alarm is displayed with a serial number • M19 has been programmed, although this function is not implemented in the control
Remedy:	• Check part program or PLC user program • Check NC MD • Have option checked by Service and retrofit if necessary

c) *Up to and including SW 6*

229*c)	Parameter assignment error
Scan:	• During ramp-up • On warm restart • On MD changes
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • A service number is shown in the diagnostics display of the spindle concerned
Explanation:	Errors are ascertained on checking the spindle machine data or the associated C axis
Remedy:	Check the following machine data and correct if necessary: • MD 155 • MDs 160 to 163 • MD 168 • MD 200*, MD 384* • MD 364*, MD 368* • MD 400*, MD 460* • MDs 403* to 410* • MDs 419* to 426* • MDs 478* to 485* • MD 435* • MD 455*, MD 456* • MD 458* • MD 459* • MD 461* • MD 463* • MD 466* • MD 468* • MD 1800* A service number is shown in the diagnostics display of the spindle concerned. You can see the meaning of the service number from the ”Extended Spindle Functions” Function Manual.

c) Up to and including SW 6

230*c)	Function cannot be executed
Scan:	• On programming the M function for selecting or deselecting C axis operation • On programming the G function for selecting or deselecting synchronous operation
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	C axis operation or synchronous operation has been selected or deselected for a spindle which is defined on a 16 bit servo CPU
Remedy:	• Check assignment of spindle and axis to the servo CPU (MD 400*, 200*, 461*) • Check the part program

231*c)	Illegal NC command
Scan:	On selecting M19
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	M19 has been requested for a spindle which is in C axis or synchronous operation
Remedy:	• Check the part program • Check the PLC program (M19 of PLC)

c) *Up to and including SW 6*

2410	**FA1 in controlled coupling**	
2411	**FA2 in controlled coupling**	
2412	**FA3 in controlled coupling**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Only the actual values of the LA/LS are used for determining the set-points of the FA/FS. • The coupling between LA/LS and FA/FS is maintained for the time being	
Explanation:	• The alarm is displayed with a serial number • An error of the LA/LS or other axes or spindles of the same mode group has occured. The coupling is maintained as long as the FA/FS itself is not faulted. This is only guaranteed for a certain time which is specified in MD 956/976/996.	
Remedy:	Clear fault with the LA/LS or the concerned axis or spindle within the mode group. The coupled axes/spindles should be separated.	

2413	**FA1 Emergency retraction**	
2414	**FA2 Emergency retraction**	
2415	**FA3 Emergency retraction**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and emergency retraction monitoring has been enabled	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The emergency signal EMERGENCY RETRACTION ACTIVE is set • When using the mixed I/O module, a respective hardware signal is set	
Explanation:	• The alarm is displayed with a serial number • The synchronism between LA/LS and FA/FS is continuously monitored. If the deviation exceeds the emergency retraction threshold (MD 954/974/994), the alarm is activated if emergency retraction monitoring has been enabled (emergency signal or SD bit).	
Remedy:	• Is emergency retraction enable (emergency signal of SD bit) desired? • Check drive of FA/FS and LA/LS • Check limiting values for speed and acceleration • Check emergency retraction threshold • Check the transmission ratios	

2416	**FA1 Speed limitation**	
2417	**FA2 Speed limitation**	
2418	**FA3 Speed limitation**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected	
Effect:	• Processing interruption • NC START disabled • The speed of the FA/FS is limited to its maximum value • The setpoint determined by the balancing controller is ignored during limitation	
Explanation:	• The alarm is displayed with a serial number • The setpoint calculated for the FA/FS has exceeded the maximum speed of the FA/FS. Synchronism between LA/LS and FA/FS is at risk.	
Remedy:	• Reduce the speed of the LA/LS • Check limiting values for the speed of the FA/FS • Check the transmission ratios	

2419	**FA1 Acceleration limitation**	
2420	**FA2 Acceleration limitation**	
2421	**FA3 Acceleration limitation**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and the MD bit ”Suppression of acceleration limitation ” is not set	
Effect:	• Processing interruption • NC START disabled • If the ”Acceleration limitation synchronous” signal (emergency signal, SD bit) is set, traversing is continued at maximum acceleration. Set part positions suppressed during the movement are subsequently traversed to. • The setpoint speed determined by the balancing controller is ignored during limitation to maximum acceleration	
Explanation:	• The alarm is displayed with a serial number • The acceleration calculated for the FA/FS has exceeded the maximum acceleration of the FA/FS. Synchronism between LA/LS and FA/FS is at risk.	
Remedy:	• Reduce the speed or the acceleration of the LA/LS • Check the limiting value for acceleration of the FA/FS • Check the transmission ratio • Suppress acceleration limitation if necessary (MD bit)	

2422	**FA1 Configuration error**
2423	**FA2 Configuration error**
2424	**FA3 Configuration error**
Scan:	• During ramp-up • On warm restart • On MD changes
Effect:	• NC START disabled • The incorrectly configured LA/LS is ignored
Explanation:	• The alarm is displayed with a serial number • The following checks are made: – Are the real LA/LS assigned to a measuring circuit? – Is the FA/FS assigned to a measuring circuit? – Is one axis/spindle defined as FA/FS more than once? – Has the FA/FS itself as LA/LS? – Is there any concatenation of LA/LS and FA/FS?
Remedy:	• Check the measuring circuit assignment • Configure LA as a fictitious LA or assign the LA to a measuring circuit • Check the configuration of LA/LS and FA/FS

2425	**FA1 Suppression inhibited**
2426	**FA2 Suppression inhibited**
2427	**FA3 Suppression inhibited**
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and a suppressing travel command for the FA/FS is given
Effect:	• Processing interruption • NC START disabled • The suppressing traversing movement of the FA/FS is not performed • The coupling is maintained
Explanation:	• The alarm is displayed with a serial number • A suppressing traversing movement of the FA/FS is only possible if suppression has been enabled (Emergency signal or SD bit). Suppression must also be enabled for the functions: – ”In-process synchronisation” (ELG) – ”Semi-automatic centering” (ELG) – ”Defined angle offset” (synchro-spindle)
Remedy:	• Set emergency signal or SD bit ”FA/FS suppression On” or • Avoid suppression

3000	**General programming error**
Scan:	On processing a part program or program block in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • In one block of the program a general, not precisely definable, programming error has been made • Division by 0 • A non-existent G function has been programmed • Non-existent R parameters have been programmed • No +, -, **/**, * has been programmed with R parameter chaining • Value range exceeded with T parameter calculation • Number of decades exceeded (M, S, T, D, H, L, P, F) • Number of subroutine passwords P not programmed directly behind L • Main block":" programmed in the subroutine • 2 decimal points programmed • Decimal point programmed with M, S, T, D, H, L, P • More than 8 decades programmed
Remedy:	• Check the faulty block in "Correction block" display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3001	**Geometry parameters > 5**
Scan:	On processing part programs
Effect:	• Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • More than 2 radii or 2 angles have been programmed in the block • More than 5 geometry parameters, such as axes, interpolation parameters, radii, angles ... have been programmed in the block
Remedy:	• Check the faulty block in "Correction block" display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3002	**Polar/radius error**
Scan:	On processing part programs
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • In the block with polar/radius programming, no programming for: – Angle – Radius • Full circle – Coordinates for the centre point not specified as absolute values
Remedy:	• Check the faulty block in “Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3003	**Invalid address**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • A different address has been programmed from that defined in NC MD or the plane assignment (cube) and the axis definition in the channel do not agree Example: Incorrect: `N20 G0 C100 LF` (rapid traverse 4th axis) In NC MD the 4th axis has been defined with address Q Correct: `N20 G0 Q100 LF` • The programmed axes are not defined in the selected mode group (with G200)
Remedy:	• Check the faulty block in “Correction block” display • Check the channel-specific MD 548* • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3004	CL800 error
Scan:	Cyclic
Effect:	Processing interruption
Explanation:	Alarm is displayed with the channel and block number Formal errors • wrong characters entered (0...9, a...f are permitted) • @-number greater than 3 decades • @-number or @-function not implemented in SINUMERIK 880 • @-number or @-function not programmable or defined in CL800 Input error with address letters and numerical values • Incorrect address letters (K, R, P are permitted) • Number of decades too high (max. K 8 decades permitted) • Number of decades too high (max. R 4 decades permitted) • Number of decades too high (max. P 4 decades permitted) • R parameter number not defined or too high • Point programmed in R parameter number • Point programmed in P parameter number • Incorrect number of words Input error with specific @-functions: Program branching • Error in the block number (programmed point, block number greater than 4 decades) Data transfer system cell - R parameter • Constant or R parameter contents programmed too large for information such as: axis number, channel number, TO area, NC/PLC machine data, NC setting data, D number, P number, group for zero offsets, default "COARSE/FINE" alarm number • Bit number too large (0 to 7 are permitted) • System cell non-existent • Incorrect value input for system cell Mathematical and logical functions • Value selected too high for square root (+ 00 000 001...99 999 999 permitted) • Incorrect angle selected for sine (-360(0) +360 permitted) • Two constants used for: Angle from two vector components, OR, EXOR, AND, NAND • Incorrect sign input for logical functions (0, 1 permitted) (only bits and bytes) (max. 8 bits) NC-specific functions • Incorrect address letter used for number of axes • Number of axes selected too high (max. three axes permitted per block) • No axis name programmed (0)

3005	Contour definition error
Scan:	On processing in AUTOMATIC mode
Effect:	• Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • The coordinates in blueprint programming have been defined so that no intersection point is given • Too many geometry values have been programmed
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3006	Wrong block structure
Scan:	On processing part programs in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Approach to reference point in the program and specification of wrong G function or more than 1 axis or an illegal axis • Incorrect thread lead parameters with G33/G37 • More than 2 axes programmed with G33/G34/G35 • More than 3 M functions in the block • More than 1 S function in the block • More than 1 T function in the block • More than 1 H function in the block • More than 4 auxiliary functions in the block • More than 6 axes+geometry parameters • More than 5 axes with G00/G01/G02/G03/G200 • More than 2 axes with G10/G11 • More than 2 axes with G02/G03 (circle radius programming) • G04 programmed with addresses other than X, F or S • Not only G04 programmed in the block • M19 S programmed with other functions • Incorrect circular parameters with G02/G03 axes • Axis missing with circle radius programming • G10/G11 block must be programmed in front of the 1st G110 block in the program • G110 must not be programmed with axes • Incorrect programming in conjunction with G176 freezing function • G200 programmed with other functions • A traversing movement has been programmed with G91 immediately after a block with G74 (Reference point approach in program) or with G130/G230/G330 (Transformation) or with @720 (Measuring). Only G90 is permissible. • With cylindrical interpolation active, circular contours cannot be programmed with interpolation parameters
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3007	**Wrong setting data program**
Scan:	On processing program blocks in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • M19 without S word • Spindle not available • Illegal setting data, e.g. G25, G26, G92 X. . . Y. . ., G92 D. . ., T. . ., I. . . or J. . .
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3008	**Subroutine error**
Scan:	On processing programs in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • M17 not in subroutine • M02/M30 in subroutine • M17 in subroutine • More than 3 subroutine levels • MD 284 = 0 with G203/G204
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3009	**Program disabled**
Scan:	• On NC START • When editing a program during processing
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • A program is called with NC START which has been disabled by ”Opening”, "Copy" or "Rename". A program being edited may not be called with NC START. • If a program is to be edited while another program is being processed it must be disabled
Remedy:	On completion of editing the disabled program must be enabled

3010	**Intersection error**
Scan:	On processing part programs in AUTOMATIC mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • An error has been ascertained during reference preparation in conjunction with calculation of the intersection point caused by: – Contour program without G00, G01, G03 – Contour program with "Clear buffer memory" (@ 714) – Programmed axes different from selected plane – No intersection found – Circular cutting path – R parameter number non-existent – Incorrect plane assignment
Remedy:	Check program containing the contour

3011	**Axis twice/axis number > 5**
Scan:	On processing part program blocks in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • An axis has been programmed twice in the same block • More than five axes have been programmed
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3012	**Block not in memory**
Scan:	On block search or jumps in the part program
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • The block number has not been found in the program in a block search • In a jump in the program the programmed block number could not been found • The program has not been terminated with M30, M17
Remedy:	• Check the part program for correct block number destination or correct program termination with M30/M02 or M17

3013	**Simulation data not enabled**
Scan:	While simulation is active
Effect:	Simulation is stopped
Explanation:	• Alarm is displayed with the channel and block number • Data have been accessed in the simulated part program that are disabled for simulation
Remedy:	• Check program • Use R0 to R199 with @ commands instead of R900 to R999 • Check NC MD 5009 bit 4

3014	**Axis blocked (disabled) in channel**
Scan:	On processing program blocks in AUTOMATIC or MDA mode or when processing external programs is specified
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • The programmed axis has been disabled for this channel via NC MD 576*
Remedy:	• Pay attention to the machine tool manufacturer's Programming Instructions • Correct machine data if necessary

3015	**Main block not in memory**
Scan:	On automatic block search
Effect:	Machining is not started
Explanation:	• Alarm is displayed with the channel and block number • No main block is found in front of the target block during automatic block search
Remedy:	• Check block found • Use different block search

3016	**External data input error**
Scan:	On data input from PLC to NC
Effect:	Processing interruption
Explanation:	• The alarm is displayed with a serial number • For external data input from PLC and NC – The code is incorrect – The value is too large – The dimension identifier is invalid
Remedy:	• Check PLC program

3017	**Part program number occurs twice**
Scan:	On POWER ON
Effect:	Processing interruption
Explanation:	• The alarm is displayed with a serial number • There is a part program present on the UMS submodule which is also present in the part program memory of the NC (with the same identifer)
Remedy:	• Delete the program in the part program memory or rename • Use a different UMS

3018	**Distance to contour too large**
Scan:	On AUTO interruption in contour block and departure from interruption point
Effect:	• Processing interruption • Supplementary alarm 2048
Explanation:	• Alarm is displayed with the channel and block number • This alarm is triggered when automatic operation is interrupted while a contour block is being executed and when the axes are positioned in an area, e. g. for tool change, that is outside the permissible tolerance for repositioning (scratch) (MD9). In order to avoid wrong positioning, the supplementary alarm 2048 (circle point error) is set. • Repositioning is only possible after Reset
Remedy:	• Cancel alarms and activate block search to the interruption point • Check whether MD 9 value can possibly be increased

3019	**Option RS 232 not available**
Scan:	On selection of interface
Effect:	Function is not executed
Explanation:	• Alarm is displayed with the block number • The 2nd RS 232 C (V.24) interface has been started either from the PLC or with the softkey, but the option is not fitted
Remedy:	• Transfer data via the 1st RS 232 C (V.24) interface • Have option checked by Service and retrofit if necessary

3020	Option not available
Scan:	On specifying a function that has not been implemented
Effect:	The function is not executed
Explanation:	• Alarm is displayed with the block number • A function has been programmed or selected that is not implemented in the control
Remedy:	Have option checked by Service and retrofit if necessary

3021	CRC/TNRC contour error
Scan:	With CRC/TNRC selected NOT: – in selection block – in deselection block
Effect:	MD 5024.0=0: Machining is not interrupted, alarm can be acknowledged MD 5024.0=1: Machining is not interrupted. The operator can either stop machining (NC RESET) or acknowledge the alarm and then continue machining with NC START
Explanation:	• Alarm is displayed with the channel and block number • The compensation calculation results in a traversing movement which is opposite to the one programmed • No intersection is obtained with linear-circular or circular-circular transitions in the angle range > 180°
Remedy:	Check programming and tool radius

3022	More than one spindle programmed
Scan:	On processing part programs in AUTOMATIC or MDA mode
Effect:	The function is not executed
Explanation:	• The alarm is displayed with a serial number • Only one spindle can be programmed in one part program block
Remedy:	Split spindle programming into two or more blocks

3024	**Display description not available**
Scan:	On display selection
Effect:	The display selected does not appear
Explanation:	• The alarm is displayed with a serial number • An attempt has been made with a programmed softkey to select a display that does not exist in the UMS or system memory
Remedy:	Check the configured display number and the configured softkey function using the programming workstation

3025	**Display description error**
Scan:	On display selection
Effect:	The display selected does not appear
Explanation:	• The alarm is displayed with a serial number • A display has been programmed with graphic elements although the control has no "Graphics" option • The display selected has too many variables or fields • A type of display has been programmed which the control does not recognize
Remedy:	• Check the display using the programming workstation • Have option checked by Service and retrofit if necessary

3026	**Graphics/text too voluminous**
Scan:	On display selection via softkey
Effect:	The display contents is not displayed completely
Explanation:	• The alarm is displayed with a serial number • Configuring error in the display selected • Sum of graphics and text elements too large
Remedy:	• Check the display using the programming workstation • Split the contents over two displays if necessary

3027	**Graphics command too voluminous**
Scan:	On display selection via softkey
Effect:	No graphics display
Explanation:	• The alarm is displayed with a serial number • Sum of the graphics commands in display selected too large
Remedy:	• Check the display using the programming workstation • Split the contents over two displays if necessary
Caution:	This alarm causes alarm 3026 to be triggered

3028	Too many fields/variables
Scan:	On display selection via softkey
Effect:	The display selected does not appear
Explanation:	• The alarm is displayed with a serial number • Programming error in the display selected. The number of fields or variables is limited. A maximum number of fields/variables cannot be specified as the fields/variables may have differing formats and lengths
Remedy:	• Check the display using the programming workstation • Reduce the number of fields/variables • Split the contents over two displays if necessary

3029	Graphics option not available
Scan:	On display selection via softkey
Effect:	No graphics display
Explanation:	• The alarm is displayed with a serial number • A display has been programmed with graphic elements although the control has no "Graphics" option
Remedy:	• Program displays without graphic elements • Have option checked by Service and retrofit if necessary

3030	Cursor memory not available
Scan:	On display selection via softkey
Effect:	The display selected is treated as if no cursor were available
Explanation:	• The alarm is displayed with a serial number • The cursor memory programmed for the display selected is not correct (number not allowed or too large)
Remedy:	Redefine the cursor memory using the programming workstation
Caution:	The function of the cursor memory is to position the cursor where it was previously when the display is called again

3032	**Too many fields/variables**
Scan:	Cyclic
Effect:	Not all variable values/texts are displayed
Explanation:	• The alarm is displayed with a serial number • The programmed variable display section of the display description is too voluminous
Remedy:	• Check the display using the programming workstation, reconfigure if necessary • Reduce variable display part

3033	**Display text not available**
Scan:	Cyclic
Effect:	The display text does not appear
Explanation:	• The alarm is displayed with a serial number • The display text prepared at the programming workstation has not been transferred to the link list
Remedy:	Check link list and re-link using the programming workstation. Watch out for linking errors.

3034	**Text not available**
Scan:	On display selection via softkey
Effect:	The special text is not displayed
Explanation:	• The alarm is displayed with a serial number • In the display selected the following texts were either linked incorrectly or not linked: – Menu texts – Interactive dialog texts – Mode texts – Alarm texts
Remedy:	Check the display using the programming workstation and reconfigure if necessary

3035 Ind. addressing incorrect

Scan: Cyclic

Effect: Variable values/texts are not displayed

Explanation:

- The alarm is displayed with a serial number
- The display description for indirect addressing is not correct. This refers to the specifications in the display heading for:
  - Data group
  - Data type
  - Data number
  - Data block
- Specified number of indirect elements of list/display (EL) is not correct
- Variable text (VT) selected, but status not deselected or offset not deselected
- Variable value (VV) selected, but status not deselected

Remedy:

- Check specifications for
  - Range start pointer
  - Range length pointer
  - Cursor

  and check their interrelationship
- Check deselection of status or offsets

3036 Variable status incorrect

Scan: Cyclic

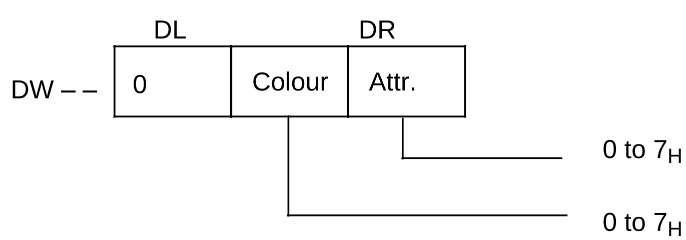
Effect: The status indicated in the display description is displayed

Explanation:

- The alarm is displayed with a serial number
- An illegal status is read

  Illegal: Low Nibble (bits 0 to 3) > 7_H
  High Nibble (bits 4 to 7) > 7_H
- An illegal display status is specified by the PLC

Remedy: The status specified by the PLC user program must be corrected



3040	Field/variable not displayable
Scan:	Cyclic
Effect:	Not all variable values/texts are displayed
Explanation:	• The alarm is displayed with a serial number • Field/variable incorrectly configured (data group not available) • Field/variable configured with too few characters • Field/variable overflow (value range exceeded) • Format error, format cannot be changed • Illegal pointer • If the alarm occurs with standard displays, the value range has been exceeded
Remedy:	Check field/variable using the programming workstation, delete and re-enter if necessary

3041	Too many fields/variables
Scan:	Cyclic
Effect:	Not all variable values/texts are displayed
Explanation:	• The alarm is displayed with a serial number • The display description has been made too voluminous, the internal buffer memory is not large enough to process the display • Too many variable values or texts have been programmed • A maximum number of fields/variables cannot be specified as the fields/variables have differing formats and lengths
Remedy:	The desired information must be reduced or split over several displays using the programming workstation

3042	Display description error
Scan:	On display selection via softkey
Effect:	The display selected does not appear
Explanation:	• The alarm is displayed with a serial number • An error has been found in the display description which cannot be exactly defined, e. g. a non-existent field has been programmed (NC MD for 33rd axis with 880)
Remedy:	Check the display using the programming workstation

3044	Graphics subroutine too large/missing
Scan:	Cyclic
Effect:	No variable values/texts are displayed
Explanation:	• The alarm is displayed with a serial number • An error has occurred in the system program or graphics processor as regards the variable display design
Remedy:	Replace system program

3045	Graphics subroutines used too large
Scan:	On display selection via softkey
Effect:	No graphics display
Explanation:	• The alarm is displayed with a serial number • The total of the graphics subroutines used in the display is too high
Remedy:	• Simplify display description • Use arcs instead of polygon curves

3046	Variable faulty
Scan:	On display selection via softkey
Effect:	Not all variable values/texts are displayed
Explanation:	• The alarm is displayed with a serial number • A variable text **without** end identifier has been programmed in display description caused by an error in the programming workstation software. A transfer format error is the result.
Remedy:	Check programming workstation (error must be located at the interruption point of the following elements that are no longer displayed)

3050	Wrong input
Scan:	During the simulation function, when reading tool offsets
Effect:	The function is not executed
Explanation:	• The alarm is displayed with a serial number • Number of tool probes too small/large • Tool type > 39 or < 0 (possible values: 0 to 39) • Empirical value number too large • Measuring parameter > 2 or < 0 (possible values: 0 to 2)
Remedy:	Check simulation data or tool offsets and correct values if necessary

3051	Programmed feed missing/wrong
Scan:	While simulation is active
Effect:	Simulation is not stopped
Explanation:	• Alarm is displayed with the channel and block number • For graphic simulation a program is simulated which does not contain a feed value in front of the 1st traversing block
Remedy:	Correct program

3052	Simulation error
Scan:	While simulation is active
Effect:	Simulation is not stopped
Explanation:	• Alarm is displayed with the channel and block number • Internal simulation error
Remedy:	Have error removed by Service

3054	Prog. not simul. correctly
Scan:	While simulation is active
Effect:	None, simulation continues
Explanation:	• Alarm is displayed with the channel and block number • A function has been programmed that cannot be properly run in simulation – Programmed plane G16 does not agree with a standard plane (G17/G18/G19) – Synchronisation has been lost with dual saddle simulation by means of G04 – G06/G68/G74 cannot be simulated – Coupled motion cannot be simulated – Transformation cannot be simulated
Remedy:	None

3058	**VTE 340 cannot be aborted**
Scan:	Each time terminal emulation is deselected
Effect:	Terminal emulation cannot be terminated in this mode (return to NC operation not possible)
Explanation:	• The alarm is displayed with the block number • Exit from emulation is not permitted while it is in setup mode , in graphics mode or in local edit mode
Remedy:	• Exit from setup mode of the terminal emulator • Exit from graphics mode of the terminal emulator • Exit from local edit mode of the terminal emulator

3059	**Setup cannot be called**
Scan:	Each time setup mode is selected
Effect:	Setup mode cannot be selected within terminal emulation
Explanation:	• The alarm is displayed with the block number • Setup mode must not be selected as long as the terminal emulation is in graphics mode
Remedy:	Exit from the graphics mode of the terminal emulation

3060 d)	**VTE 340 cannot be called**
Scan:	With Teleservice and terminal emulation
Effect:	Terminal emulation not possible
Explanation:	• The alarm is displayed with a serial number • Incorrect interface was parameterized when terminal emulation was called • With TELESERVICE active, an attempt is made to switch to terminal emulation
Remedy:	• Parameterize correct interface

d) From GA2, SW 1

3061 d)	**Fault in keyboard**
Scan:	On POWER ON and cyclic
Effect:	The operator panel keyboard and any connected full keyboard are out of action
Explanation:	This error message occurs if, for example, the reset key on the keyboard interface module is pressed or if the connection between the OPI and INTKEY modules is faulty.
Remey:	Check connection between OPI and INTKEY cards, then execute control POWER ON routine

3062 d)	**No outside line**
Scan:	When setting up connection (Teleservice)
Effect:	No connection can be set up at present
Explanation:	• The alarm is displayed with a serial number • All outside lines are currently busy • Adapter not connected to post-office network
Remey:	Try to set up connection again after a delay

3064 d)	**Adapter not reached**
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • Adapter not recognized
Remey:	• Check RS232C (V.24) connection between control and adapter • Check adapter power supply • Use of adapter that has not been released

d) From GA2, SW 1

3066[d)]	Teleserv. cannot be called
Scan:	When calling the Teleservice
Effect:	Teleservice cannot be implemented
Explanation:	• The alarm is displayed with a serial number • The TELESERVICE function is a specified device type for the serial interfaces of the operator panel. This device type must be entered in the relevant setting data. • The TELESERVICE function was interlocked by the key-operated switch.
Remedy:	• Enter TELESERVICE (11H) device type in the setting data of the relevant serial interfaces for reading in and out • Cancel interlock with key-operated switch

3067[d)]	No phone number
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • No number entered in subroutine compiled by user • Non-compliance with brackets in subroutine compiled by user
Remedy:	• Check number entry in subroutine • Check brackets that enclose the number

3068[d)]	Phone number memory missing
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • No subroutine number entered in the general machine data that contains this number with the phone numbers • Non-compliance with brackets in subroutine compiled by user
Remedy:	• Enter subroutine number specified by user in the general machine data

d) From GA2, SW 1

3069[d)]	Teleservice transm. error
Scan:	When calling the Teleservice
Effect:	Teleservice cannot be called
Explanation:	• The alarm is displayed with a serial number • Parameter settings of the two adapters are different • Non-compliance with correct sequence when calling Teleservice
Remedy:	• Check parameter settings of the two adapters • Eliminate operator error (deactivate Teleservice) and acknowledge alarm

3072	Message not available
Scan:	Cyclic
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Alarms are provided in cycle preparation for which no text has been programmed • NC alarms occur for which no text is provided in the system • Study list of alarm displays and check for alarm numbers without text
Remedy:	• Program text for cycle alarms • Send a message to the system management in the event of system alarms

3077[d)]	Wrong rotary axis
Scan:	With each block change
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • With G203/G204 a rotary axis was programmed without assignment to a spindle
Remedy:	Assign the axis number of the rotary axis to a spindle with MD 461*

d) From GA2, SW 1

3078[d]	**Wrong infeed axis**
Scan:	With each block change and in selection block for tapping
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • Infeed axis not contained in plane G16-G19
Remedy:	Infeed axis must be perpendicular to current plane

3079[d]	**No S value for thread**
Scan:	With each block change and in selection block
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • No S value programmed in selection block G203/G204
Remedy:	Program S value

3081	**CRC/TNRC not selected for approach**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• NC START disabled • The program is executed without approach element
Explanation:	• Alarm is displayed with the channel and block number • CRC/TNRC (G41/G42) has not been selected in or in front of the approach block • CRC/TNRC has not been deselected in or after the approach block
Remedy:	• Soft approach/retraction is only possible when CRC/TNRC has been selected since otherwise the programmed compensating movement cannot be precisely calculated • Correct program (G41/G42) • Program D0 with G41/G42 if necessary

d) From GA2, SW 1

3084	Illegal working area limitation
Scan:	Cyclic
Effect:	The control automatically enters the maximum value permitted by the traversing range in the working area limitation
Explanation:	• Alarm is displayed with the block number • A value has been entered in the minimum or maximum axis-specific working area limitation that lies outside the permissible traversing range of the relevant axis
Remedy:	• Check input • Check program (G25/G26, @...) • Look up maximum traversing range in table (combination of axis-specific position control resolution and input resolution)

3085	NC CPU time watch-dog
Scan:	Cyclic
Effect:	None, brief irregular axis motions may occur
Explanation:	• Alarm is displayed with the block number • Time overload in the NC CPU as a result of parallel program operation or due to selected functions such as – Transformation (TRANSMIT, ...) – Coupled motion
Remedy:	Stop program with NC STOP

3086	Illegal transformation selection
Scan:	On selection/deselection of transformation via the command channel from the PLC
Effect:	NC START disabled
Explanation:	• The alarm is displayed with a serial number • The PLC has transferred an illegal value via the command channel. This is evaluated in the error byte in the PLC.
Remedy:	Check PLC user program

3087	**Error in transformation data**
Scan:	• On POWER ON • On warm restart
Effect:	The transformation data block selected is disabled
Explanation:	• Alarm is displayed with the block number • An illegal value has been assigned to one NC MD in the transformation data block selected. The illegal NC MD number is entered in the block number of alarm 3087
Remedy:	Check NC MD for transformation data (block number of alarm 3087)

3089	**K_V factor for thread too small**
Scan:	When thread is programmed
Effect:	Calculation is terminated if the calculated K_V factor exceeds the limit under consideration of the IPO time
Explanation:	• The alarm is displayed with a serial number • The input K_V factor is too small
Remedy:	Enter larger K_V factor, also refer to Installation Instructions, Function Description "Extended Thread Package".

3090	**Following axis programmed**
Scan:	Cyclic
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Alarm 3090 is output if an axis which has been defined as a folllowing axis is to be traversed in JOG mode on its own
Remedy:	• Other definition of the coupled axis grouping to prevent this axis from being a following axis • Deselect coupled motion function

3091	**Reduction at SW prelimit switch**
Scan:	In AUTOMATIC or MDA mode when processing a part program block or when positioning the axes in JOG mode
Effect:	Reduce speed to the value set in the machine data
Explanation:	• Alarm is displayed with the block number • The software prelimit switch has been overrun, the axis speed is decelerated to the reduced speed (MD 1)
Remedy:	• Check the traversing block • Check the value in NC MD 1100 (prelimit switch) • Position axis outside the prelimit switch range and cancel the alarm by acknowledging it

3092	**Programmed speed too high**
Scan:	On processing programs in AUTOMATIC or MDA mode
Effect:	None
Explanation:	• Alarm is displayed with the block number • The specified speed (either programmed or via override setting) is higher than the path speeds resulting from the maximum axis speeds
Remedy:	• Program slower path speed or check override • Check NC MD 280* (maximum speed) • Program lower spindle speed with G95

3093[d)]	**G171 illegal**
Scan:	When processing in AUTOMATIC or MDI
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the channel and block number • No FIFO assigned to current channel • FIFO Option not activated
Remedy:	• Correct program • Check MD

d) From GA2, SW 1

3094[d)]	**Error in compensation data**
Scan:	• On POWER ON • On warm restart
Effect:	IKA compensation values are not calculated
Explanation:	• Alarm is displayed with the block number • Error occurred when converting error curves for IKA • The following outputs are also made: - Channel No. contains error No. Block No. (N....) contains error location - Error No. 1: Only one of the start and end pointers is 0. Both must be 0 or <> 0. Block No. = No. of error curve - Error No. 2: End pointer not greater than start pointer Block No. = No. of error curve - Error No. 3: Within an error curve section, actual position (n + 1) is less than actual position (n) Block No. = No. of compensation point - Error No. 4: Within an error curve section, actual position (n + 1) equals actual position (n) Block No. = No. of compensation point - Error No. 5: Thread pitch within an error curve section is 3 45 degrees Block No. = No. of compensation point **Example:** Alarm line: "`3094 3 N105` Error in compensation data": At compensation point 105, actual position 106 is less than actual position 105.
Remedy:	• Check and correct IKA data • Implement POWER ON start or warm restart

3095[d)]	**ISDN error**
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • Adapter not connected to ISDN socket via S0 connector
Remedy:	• Check adapter S0 connector • Execute connection setup routine again

d) From GA2, SW 1

3096d)	User busy
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • The user's line is busy
Remedy:	• Try to set up connection again after a delay

3097d)	User cannot be reached
Scan:	When setting up connection (Teleservice)
Effect:	Connection setup impossible
Explanation:	• The alarm is displayed with a serial number • The user (Service Department) does not respond; adapter not in auto-answer mode
Remedy:	• Service must switch an adapter to the auto-answer mode before the connection is setup

3098d)	VTE communication error
Scan:	Cyclic, with each data transfer from VT 340 to the computer
Effect:	Faulty RS232C (V.24) connection between VT 340 and computer
Explanation:	When transferring data (VT 340 to computer), an error that prevents the transmission occurs (e.g. cable not inserted). All subsequent transfers to the computer are suppressed until the problem has been eliminated.
Remedy:	Check RS232C (V.24) communication program in computer and RS232C (V.24) connection between VT 340 and computer

d) From GA2, SW 1

3150	**No reaction from PLC**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Alarm 3150 appears after approx. 0.5 to 2 seconds if the PLC to be cancelled by overall reset is not fitted or if a selected PLC fails during the overall reset operation
Remedy:	• Overall reset of correct (existing) PLC • Replace PLC

3151	**Interface occupied by PG**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Alarm 3150 appears if data are just being transferred between the programmer and one of the PLCs
Remedy:	• Abort programming function • Terminate programming function

3152	**Incorrect PLC mode**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Wrong operation mode set at the PLC
Remedy:	Set correct operating mode (cycle/stop)

3153	**Funct. ident. unknown in PLC**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The correct function identifier is unknown to the PLC
Remedy:	Select correct function

3154	**Error in PLC program memory**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • Error in the PLC program memory
Remedy:	Report to Service

3155	**PLC RAM not inserted**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The PLC RAM is not inserted
Remedy:	Insert PLC RAM

3156	**Function disabled or active**
Scan:	On function start via softkey
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The PLC function is disabled because another function is active
Remedy:	Avoid simultaneous signal status display at the programmer

3157	**Halt in thread**
Scan:	On thread cutting
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the block number • During thread cutting a halt has occurred in revolutional feed which has damaged the thread
Remedy:	Check axis-specific feed disable (DB 32)

3158	PLC not available
Scan:	On selection via softkey in a programmed display
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The programmed field refers to a non-existent PLC number specification
Remedy:	Check assignments using the programming workstation and correct

3159	Data block not available
Scan:	On selection via softkey in a programmed display
Effect:	None
Explanation:	• The alarm is displayed with a serial number • The programmed field refers to a non-existent data block
Remedy:	Check assignment using the programming workstation and correct

3160	Tool simul. w/o dimensions
Scan:	While simulation is active
Effect:	Simulation is stopped
Explanation:	• Alarm is displayed with the block number • The programmed T number does not exist in the simulation tool data
Remedy:	Check the programmed T number in the simulation tool data and enter new values if necessary

3161	Tool form simulation not possible
Scan:	While simulation is active
Effect:	The simulation is stopped
Explanation:	• Alarm is displayed with the block number • An illegal tool form is defined in the simulation tool data under the programmed T number
Remedy:	• Define other tool form • Check program

3162 Not all programs deleted

Scan:	On reading-in via the serial interface
Effect:	None
Explanation:	• Alarm is displayed with the block number • An entire program section is to be deleted with %CLF on reading-in via the serial interface. For this, a program number has been specified which is just being processed and thus cannot be deleted
Remedy:	Select ”Clear program” with %CLF via the serial interface once again after the program concerned has been completed

3163 Softkey start not possible

Scan:	On starting a subroutine via softkey
Effect:	The subroutine is not started
Explanation:	• The alarm is displayed with a serial number • The subroutine to be started in a defined channel cannot be started via softkey
Remedy:	Check whether • Channel is available • Subroutine with @00f in 1st block • Channel is already assigned (program running)

3164 Axis converter error

Scan:	• On processing in AUTOMATIC or MDA mode • On graphic simulation
Effect:	Processing interruption

Explanation:	• Alarm is displayed with the channel and block number
	• Incorrect input in the axis conversion list (SD)
	  – Axis name not entered
	  – Axis name does not exist
	  – Axis fictitious (for transformation)

Remedy:	Correct axis conversion list

3165[d)]	**Remote PG failure**
Scan:	Cyclic when remote PG is selected
Effect:	None
Explanation:	The alarm is displayed with a serial number
Remedy:	• Check CP231 • Check software version for CP231 • Clear dualport RAM for CP231 if applicable (start-up mode)

3166	**Program coordination wrong**
Scan:	On processing in AUTOMATIC mode
Effect:	Machining is stopped in the event of grave errors by the PLC (Read-in disable)
Explanation:	• Alarm is displayed with the channel and block number • The channel specified in the program coordination command WAIT or INIT does not exist • The channel specified in the program coordination command START does not exist or is disabled for processing
Remedy:	• Check program coordination command in part program • Check all associated channels with a WAIT M command

3167	**Reassign T/H word**
Scan:	Each time the T/H word is rearranged
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • The part program can be used to transfer T and H words from the NC channel in which they have been programmed (source channel) to another NC channel (target channel). Each reassignment must be acknowledged by the user in the target channel before the next reassignment is made • Possible errors: – Target channel not enabled via machine data (signal from/to channel) – Reassignment in the target channel has not yet been acknowledged by the user
Remedy:	• Match part and user program • Enable target channel by machine data

d) From GA2, SW 1

3168	**Coord. not reproducible**
Scan:	On processing in AUTOMATIC mode
Effect:	None
Explanation:	• Alarm is displayed with the block number • Programming error in T/H word reassignment
Remedy:	Match part and user program

3170	**FA1 KUE wrong**
3171	**FA2 KUE wrong**
3172	**FA3 KUE wrong**
Scan:	On calculating the transmission ratios for an ELG or a sychro-spindle pair
Effect:	• Command ”Activate Kü” disabled (emergency signal or SD bit) • The old transmission ratio is maintained
Explanation:	• The alarm is displayed with a serial number • An error has occurred in the calculation of the transmission ratio. Possible causes are: – Denominator= 0 – Result too large (> 1000) – Format overrun caused by unfavourable selection of increment or axis-specific resolutions
Remedy:	• Check transmission parameters • Check increment weighting and axis-specific resolutions

3173	**FA1 Centering flank violated**	
3174	**FA2 Centering flank violated**	
3175	**FA3 Centering flank violated**	

Scan:	Cyclic when the electronic gear (ELG) is active and when the emergency signal HAE IS ACTIVE is set
Effect:	• NC START disabled • A movement beyond the already recognized flanks is suppressed
Explanation:	• The alarm is displayed with a serial number • For semi-automatic centering, the two tooth flanks are approached manually. A movement beyond the already recognized flanks is suppressed to make centering easier for the operator. The alarm is also acitvated if the interface signals HAE ON and 1ST FLANK IS APPROACHED are set without the FA being traversed in between.
Remedy:	• Withdraw FA from flank (opposite to approach direction) • Remove flank again if necessary (cancel interface signal 1ST FLANK IS APPROACHED) • Traverse at least one increment between the interface signals HAE ON and 1ST FLANK IS APPROACHED

3176	**FA1 Synch. parameter wrong**	
3177	**FA2 Synch. parameter wrong**	
3178	**FA3 Synch. parameter wrong**	

Scan:	On change of synchronisation parameters
Effect:	• NC START disabled • The last synchronisation parameters entered remain valid
Explanation:	• The alarm is displayed with a serial number • The synchronisation parameters have been incorrectly specified. The following checks are made: – Has a valid value been entered for the number of the LA? The position of the leading axis is ignored with LA number 0. – Are the values within the permissible number range (limits ±1070 000)?
Remedy:	• Enter permissible LA number (0, 1 to 5) • Enter permissible synchronisation positions

3200	Prog. coord. syntax wrong
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Syntax error: invalid command mnemonics • Invalid modification parameter, impermissible characters, too many parameters, parameters or characters missing, range limit violation by parameter
Remedy:	Correct wrong command

3201	Prog. coord. too many parameters
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • More command parameters have been programmed than allowed in the command description
Remedy:	Correct wrong command

3202	Prog. coord. range limit violation
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • One of more range parameters violate the permissible values for the upper or lower limit
Remedy:	Correct wrong command

3203	Prog. coord. illegal character
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Illegal characters occur in the coordination command
Remedy:	Correct wrong command

3204	**Prog. coord. command incomplete**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Parameter or right bracket missing in the programmed command
Remedy:	Correct wrong command

3205	**Prog. coord. R parameter error**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • An error has occured on activation of programmed R parameters
Remedy:	Correct wrong command

3206	**Prog. coord. sym. para. not allowed**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • No symbolic parameters are permitted except R parameters
Remedy:	Correct wrong command

3220	**Change G176 G175**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	The acitvated freezing function is aborted and a transition is made to the G function 175
Explanation:	• Alarm is displayed with the channel and block number • The zero offset group or the D number has been changed, an angle of rotation offset has been activated or G53 has been programmed while the freezing function for angle of rotation, zero offsets or length compensations were active. These functions cause the "Freezing" function to be terminated and the G175 function to be changed
Remedy:	Check program blocks and correct

 6ZB5 410-0HG02

3239	**Cycle exceeded by SPF**
Scan:	On reading in subroutines via RS 232 C (V.24) or file transfer
Effect:	EPROM cycle is overwritten by subroutine
Explanation:	• Alarm is displayed with the channel and block number • A subroutine has been read in whose number already existed as EPROM cycle. The EPROM cycle is no more active.
Remedy:	• The subroutine read in must be deleted if the EPROM cycle is to be maintained • If overwriting is to be prevented altogether, the setting data special bit 2, bit 3 of the relevant RS 232 C (V.24) interface or MD 5147, bit 3 in the case of file transfer must be set to 0

3240	**SPF not read in**
Scan:	On reading in subroutines via RS 232 C (V.24) or file transfer
Effect:	Subroutine has not been read in
Explanation:	• Alarm is displayed with the channel and block number • A subroutine to be read in already exists as EPROM cycle
Remedy:	If the EPROM cycle is to be overwritten, the setting data special bit 2, bit 3 of the relevant RS 232 C (V.24) interface or MD 5147, bit 3 in the case of file transfer must be set to 1

3241 d)	**M function programmed wrong**
Scan:	With each block change
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty M function programmed in block – Number of decades exceeded – Minus sign in front of equals sign (e.g.: M - 1 = 3) – M02/M03 programmed as rapid auxiliary function
Remedy:	Edit part program accordingly

d) From GA2, SW 1

3242[d)]	**S function programmed wrong**
Scan:	With each block change
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty S function programmed in block – Number of decades exceeded Dwell time > 99.9 revolutions Spindle position with oriented spindle stop > 360° Spindle value > 99999.9 – Minus sign in front of equals sign (e.g.: S - 1 = 3) – Negative sign specified
Remedy:	Edit part program accordingly

3243[d)]	**T function programmed wrong**
Scan:	With each block change
Effect:	Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty T function programmed in block – Minus sign in front of equals sin (e.g.: T-1=3) – Negative sign specified
Remedy:	Edit part program accordingly

3244[d)]	**D function programmed wrong**
Scan:	With each block change
Effect:	• Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty D function programmed in block – Number of decades exceeded – Minus sign in front of equals sign (e.g.: D - 1 = 3) – Negative sign specified – Programmed D number not in TO area of channel
Remedy:	Edit part program accordingly

d) From GA2, SW 1

3245[d)]	**H function programmed wrong**
Scan:	With each block change
Effect:	• Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty H function programmed in block – Minus sign in front of equals sign (e.g.: H - 1 = 3)
Remedy:	Edit part program accordingly

3246[d)]	**F function programmed wrong**
Scan:	With each block change
Effect:	• Processing interruption
Explanation:	• Alarm is displayed with the channel and block number • Faulty F function programmed in block – Minus sign in front of equals sign (e.g.: F - 1 = 3) – Negative sign specified
Remedy:	Edit part program accordingly

3747	**No M19 with V constant**
Scan:	In block or with each block change
Effect:	Alarm is displayed with the block number • Processing interruption • NC START disabled
Explanation:	M19 programmed with active constant cutting rate (G96 S..)
Remedy:	Deselect constant cutting rate with G97 before programming M19

4100	**No D number active**
Explanation:	CRC/TNRC selected without stating a D number within a standard cycle
Remedy:	Check part program

d) *From GA2, SW 1*

4101	**Tool radius = 0**
Explanation:	Cutter radius stated as 0 This leads to errors with standard cycles
Remedy:	Enter radius in D number

4102	**Cutter radius too great**
Explanation:	Use of this cutter would lead to contour errors with some standard cycles
Remedy:	Program different cutter

4103	**Tool too wide**
Explanation:	Grooving tool too wide for standard grooving cycle

4120	**No spindle direction programmed**
Explanation:	No spindle direction programmed before calling standard cycle
Remedy:	Edit part program

4121	**Spindle outside tolerance range**
Explanation:	If spindle speed fluctuation is excessive when using standard cycle
Remedy:	Check drive actuator, MDs

4122d)	Calculated feedrate too great
Explanation:	The feedrate calculated in the cycle for the rotary axis is greater than the maximum permissible feedrate for the axis
Remedy:	• Check and reduce the speed programmed in the S word before calling the cycle • Check whether maximum axis speed is correctly entered in machine data

4140	Machined part diameter too small
Explanation:	Too small a value entered for machined part diameter when initializing the parameters for a standard cycle
Remedy:	Check part program

4153	Thread too short
Explanation:	Overtravel compensation path calculated with L84 is greater than the thread length
Remedy:	• Program slower spindle speed • Deselect overtravel compensation • Locate plane reference point higher

4180	Option not available
Explanation:	The called standard cycle requires an Option that is not available in the control
Remedy:	Retrofit Option

4200	Check definition R (Nxxx)
Explanation:	Defining parameter incorrectly determined
Remedy:	Determine defining parameter correctly

---
d) From GA2, SW 1

5000 : 5099	**User cycle alarms**
Explanation:	Alarm called with @ 4c0 in user cycle
Remedy:	Check cycle

6000 : 9999	**PLC error message**
Explanation:	Actuation bit set in PLC user program
Remedy:	Determined by manufacturer

6000 : 9999	**PLC operational message**
Explanation:	Actuation bit set in PLC user program
Remedy:	Operational message automatically acknowledged after resetting actuation bit

1000*	**Connection assigned more than once**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Servo enable is removed after the time in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • A measuring circuit connection has been entered in NC MD 200* or 384* more than once • Alarm 58 "Parameterization error" is also issued
Remedy:	• Check NC MD 200* • Check NC MD 384*

1004* Feedrate too high

Scan: On each axis motion

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready
- Servo enable removed

Explanation:
- The alarm is displayed with a serial number
- Excessive values have been entered in MD 280* , i. e. the maximum value has been exceeded
- The limiting frequency of the C axis encoder (MD 308*) has been exceeded with a spindle operated as rotary axis (C axis)

Remedy:
- Check MD 280* and MD 155
- Check MD 308*
- Reduce feedrate or speed

1008* Speed controller limitation

Scan: Cyclic

Effect:
- Processing interruption
- NC START disabled
- Cancellation of Mode Group Ready

Explanation:
- The alarm is displayed with a serial number
- NC has switched to follow-up mode and disables rmp servo control and current control
- The speed controller has been at its limitation for longer than the time set in IAR MD 52*
- Error in parameter assignment to the ACC module
- Load moments of inertia too high
- SIMODRIVE fault
- Rotor position encoder signals to the ACC module interrupted
- Speed alarm limit reached

Remedy:
- Check parameter assignment to ACC module (IAR MD 136*; 140*; 144*; 148*)
- Reduce load moments of inertia
- Check SIMODRIVE
- Check transmission of rotor position encoder signals to the ACC module
- Reduce speed of speed controller circuit
- Increase parameters

1012*	**Parameter assignment error** [c] **Param. error drive MD** [d]
Scan:	• On warm restart • After change of MD
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • NC switches to follow-up mode and disables rpm servo control and current control • Parameter assignment error in NC or IAR machine data
Remedy:	• Check the following machine data for practical values: – IAR MD 108*[c]/MD 1396*[d] – IAR MD 116*[c]/MD 1204*[d] – IAR MD 128*[c], 132*[c]/MD 1208*[d], MD 1212*[d] – IAR MD 212*[c] – IAR MD 268*[c]/MD 1336*[d] – NC MD 160 – NC MD 264*, 268* – NC MD 364*, 368* – NC MD 1116*/MD 1682*[d]

1016*[d]	**MS switchover not poss.**
Scan:	When switching over measuring system
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Follow-up mode • Setpoint = 0
Explanation:	• The alarm is displayed with a serial number • When requesting a poisition measuring system switchover, checks are implemented to determine whether – a second measuring system is available – the actual value difference is smaller than the tolerance (MD 1216*) The alarm is triggered if one of the conditions is not fulfilled
Remedy:	• Set up second measuring system • Increase tolerance band (MD 1216*)

c) Up to and including SW 6
d) From GA2, SW 1

1024*c)	**Illegal pulse multiplication**
Scan:	• On POWER ON • After change of MD
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay • Servo enable is removed after the time in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • A value other than 1, 2, 4, 8, 16, 32, 64 or 128 has been entered in NC MD 1116*
Remedy:	Correct NC MD 1116*

1036*c)	**Actuator not ready**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • NC switches to follow-up mode and disables rmp servo control and current control • Actuator not ready
Remedy:	Check actuator (supply voltage)

1040*	**Absolute encoder error**
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Control does not start
Explanation:	• The alarm is displayed with a serial number • An absolute encoder error is scanned with ”Power On”. The error cause appears in the service status display (see also Section “Absolute encoder errors”).
Remedy:	• Check connection measuring circuit-absolute encoder • Remove cause of error
Caution:	The control must be restarted after the error has been remedied

c) Up to and including SW 6

1044*	Battery absolute submodule
Scan:	Approx. every 10 minutes
Effect:	Risk of losing absolute position on ”Power Off”
Explanation:	• The alarm is displayed with a serial number • The battery in the absolute submodule is empty
Remedy:	Replace battery under power
Caution:	**The control must not be switched off until the battery has been replaced**

1052*d)	Drive fault
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Group error with drive faults (SIMODRIVE), e.g.: local bus fault, processing time overflow, motor overtemperature, transistor fault, power circuit overtemperature, under/overvoltage, power circuit not ready, mains failure, maximum or continuous output exceeded
Remedy:	• Check drive actuator power supply • Check power consumption and heat dissipation of drive actuator • Check drive actuator hardware

1060*d)	NC command illegal
Scan:	Depending on operation
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • This alarm is triggered if the NC is not entitled to operate the drive actuator (SIMODRIVE) but wishes to – change modes (e.g. from control to axis mode) – specify setpoints (e.g. while function generator is running)
Remedy:	Obtain operation entitlement (by softkey)

d) From GA2, SW 1

1200*	**Wrong division**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • The ”Division increment from PLC” function is not possible: – NC MD 1104* (number of divisions) has invalid value – NC MD 1108* (division increment reference) has invalid value – NC MD 1112* (division increment offset) has invalid value
Remedy:	• Graduated axis has been defined as a rotary axis (NC MD 560* bit 3=1) which is not allowed • Check NC MD 1104*, 1108*, 1112* and 560* bit 3

1204*	**Traversing range limit**
Scan:	Cyclic (for linear axes only)
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • On principle, it is possible to exceed the maximum possible traversing range (defined by the combination of axis-specific position control resolution and input resolution) if no software limit switches and working area limitations are active (NC MD 560* bit 5=0). Considering, however that this would lead to traversing errors, the traversing range limit is monitored and alarm 1204* is set when the limit is exceeded
Remedy:	Return to permissible traversing range in INC or JOG mode traversing in opposite direction

1208*	**G200 Axis in more than one channel**
Scan:	When an axis is to be synchronized with G200 in one channel
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the channel and block numbers • One axis is active also in other channels with G200
Remedy:	Write part programs such that no G200 is requested with active axes

1212*	**Position behind software limit switch**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled • Programmed path is **not** traversed
Explanation:	• Alarm is displayed with the channel and block number • The programmed end point of the block is located behind the software limit switch
Remedy:	• Correct program • Check MD 244*, 228*, 232*, 236*, dependent on the PLC interface signal 2ND SOFTWARE LIMIT SWITCH ACTIVE

1216*	**Position behind working area limitation**
Scan:	On processing in AUTOMATIC mode
Effect:	• Processing interruption • NC START disabled • Programmed path is **not** traversed
Explanation:	• Alarm is displayed with the channel and block number • The programmed end point of the block is located **behind** the working area limit
Remedy:	• Correct program

1232*	**Comp. limit switch +**
Scan:	• Cyclic • With axis movement • With compensation values other than 0
Effect:	• Processing interruption • NC START disabled • Travel direction and compensation disabled
Explanation:	The effective machine position (i.e. NC setpoint - compensation value) determined by software limit switch (or working area limitation if applicable) has been reached. NC travel and compensation in this direction is disabled.
Remedy:	Retract from limit switch in JOG mode or move axis in opposite direction by changing the compensation parameters.

1236*	**Comp. limit switch -**
Scan:	• Cyclic • With axis movement • With compensation values other than 0
Effect:	• Processing interruption • NC START disabled • Travel direction and compensation disabled
Explanation:	The effective machine position (i.e. NC setpoint - compensation value) determined by software limit switch (or working area limitation if applicable) has been reached. NC travel and compensation in this direction is disabled.
Remedy:	Retract from limit switch in JOG mode or move axis in opposite direction by changing the compensation parameters.

2000*d)	**Comp. grid spac. inval.**
Scan:	Cyclic
Effect:	• NC START disabled
Explanation:	• The alarm is displayed with a serial number • Permissible range (0 ... 1000 leadscrew error compensation points) exceeded when operating drive • With leadscrew error compensation for rotary axis, a value was entered in MD 324* for the relevant axis which cannot divide 360 (degrees) into a whole number; for this reason the grid spacing is unequal
Remedy:	Change MD 324*

2001*d)	**Rpm setpoint warn limit**
Scan:	Cyclic
Effect:	• NC START disabled
Explanation:	• The alarm is displayed with a serial number • The drive oscillates if an excessive speed is specified • The digital set speed in the driving unit has exceeded the warning limit
Remedy:	• Input slower speeds • Check actual values (encoder)

d) *From GA2, SW 1*

2003*d)	Zero speed monitoring	
Scan:	• With clamping • At zero speed • With irregular deceleration • In synchronous spindle mode	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Cancellation of servo enable (on expiry of time set in MD 156)	
Explanation:	• The alarm is displayed with a serial number • When positioning, the following error could not be cancelled faster than permitted by the time entered in NC MD 156 • With clamping, the limit specified in NC MD 212* was exceeded • A mechanically clamped axis has been forced out of position • Malfunction at: – Actuator – Tacho-generator – Motor – Mechanical fault in NC measuring circuit hardware – Encoder • Incorrect feedback polarity on installation	
Remedy:	• NC MD 212* (clamping tolerance) must be greater than NC MD 204* (exact stop limit) • The time entered in MD 156 (cutout delay, servo enable) must be sufficient to allow the following error to be cancelled (only if MD 372* = 0) • The time entered in MD 372* (delay zero speed monitoring) must be sufficient to allow the following error of the individual drives to be cancelled • Check actual values (encoder) and feedback polarity	

2007*d)	Meas. loop not available	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with a serial number • In NC MD 200* or 400* the axes or spindles respectively were assigned measuring circuits that were not recognized by the NC after POWER ON • The measuring circuit module has been removed or is faulty	
Remedy:	• Check machine data • Check measuring circuit module	

d) From GA2, SW 1

2008*d)	**Control loop HW spindle**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Cancellation of servo enable (on expiry of time set in MD 337*)
Explanation:	• The alarm is displayed with a serial number • Measuring circuit differential signals are: – not in phase – short-circuited to ground – completely missing
Remedy:	• Check that the measuring circuit connector is inserted • Check measuring circuit module (insert measuring circuit shorting plug) • Check differential signals with oscilloscope • Renew encoder/cable

2009*d)	**Dirt in measur. system**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Fault reported to NC by measuring system with contamination signal
Remedy:	• Check measuring system

2011*d)	**Zero mark monitoring**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with a serial number • Pulses lost owing to transmission errors or interference, referred to one encoder revolution
Remedy:	• Check encoder pulses • Check transmission path • Switch off monitoring temporarily (HW monitor off)

d) From GA2, SW 1

2014*d)	**Rpm setpoint warn limit**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Setpoint 0 • Servo enable cancelled after expiry of time in NC MD ("cutout delay servo enable") • Follow-up mode
Explanation:	• Motor was unable to follow set speed • On installation: incorrect feedback polarity, incorrect spindle multgain Actual speed value exceeds maximum spindle speed + tolerance • Adjust spindle multgain (MD 468*) • Increase tolerance of maximum spindle speed (MD 445*) • Increase acceleration time constant (MD 419* - 426*)
Remedy:	• Check drive • Check feedback polarity • Check set speed cable • Check actual value (encoder)

2015*d)	**Spindle drift too high**
Scan:	When entering NC MD 401*
Effect:	NC START disabled
Explanation:	Entered drift value greater than 500
Remedy:	Check whether drift was correctly adjusted at driving unit

d) From GA2, SW 1

2016*[d]	**Conn. assign. 2x or more**
Scan:	After POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Cancellation of servo enable (on expiry of time set in MD 156) • Follow-up mode
Explanation:	• The alarm is displayed with a serial number • Measuring circuit connection entered more than once in MD 400* or 460*
Remedy:	• Check MD 400* • Check MD 460*

2018*[d]	**Rpm controller limitat.**
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • NC has switched to follow-up mode and disables rpm servo control and current control • External moments of inertia too high • SIMODRIVE fault has occurred
Remedy:	• Reduce external moments of inertia • Check SIMODRIVE

d) From GA2, SW 1

2019*[d]	**Param. error drive MD**
Scan:	• On runup • On warm restart • When changing MDs
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Display of a service number in diagnostics display of relevant spindle
Explanation:	• The alarm is displayed with a serial number • Faults identified when checking MDs of spindle or associated C axis
Remedy:	Check following MDs: 155, 160...163, 168, 200*, 364*, 368*, 384*, 400*, 403*...410*, 419*...426*, 435*, 455*, 456*, 458*, 459*, 460*, 461*, 463*, 466*, 468*, 478*...485*, 1800*

2020*[d]	**M19 posit. not in range**
Scan:	On selection of M19
Effect:	• Processing interruption • NC START disabled
Explanation:	• Alarm is displayed with the block number • Alarm issued if, with MDA or in a part program, M19 was programmed without an S value and the spindle setting data for oriented spindle stop contains an illegal value
Remedy:	Enter permissible value in setting data Value range 0 ... 359.99

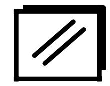
2028*[d]	**MD M19 not selected**
Scan:	• On processing AUTOMATIC or MDA mode • With external input
Effect:	Processing interruption
Explanation:	• The alarm is displayed with a serial number • "M19 S.." was initiated by PLC although the function is not enabled
Remedy:	• Check part program or PLC user program • Check MD • Have option checked by Service and retrofit if necessary

d) From GA2, SW 1

2029*d)	Drive fault
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Group error with drive faults (SIMODRIVE), e.g.: local bus fault, processing time overflow, main spindle not ready, motor overtemperature, transistor fault, under/overvoltage, power circuit not ready, mains failure, power circuit overtemperature, maximum or continuous output exceeded
Remedy:	• Check drive actuator power supply • Check power consumption and heat dissipation of drive actuator • Check drive actuator hardware

2030*d)	Spindle speed too high
Scan:	Only with MD 520* (encoder available) set
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with a serial number • Spindle speed exceeds value specified in machine data or setting data
Remedy:	• Program lower S value • Check following MDs: 403*...410*, 445*, 451* • Check PLC gear stage • Check "G92S.." with "v = constant" • Check setting data "spindle speed limitation"

d) From GA2, SW 1

2031*[d)]	**`NC command illegal`**	
Scan:	• On selection of M19 • Depending on operation	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with a serial number • NC accessed driving unit without having access right (e.g.: output of M19 for spindle in C axis or synchronous mode)	
Remedy:	• Check part program • Check PLC program (M19 from PLC)	

d) From GA2, SW 1

13 Notes on Fault Procedure

13.1 General

The section below is intended to assist in locating and rectifying possible faults in the SINUMERIK 880.

The SINUMERIK 880 GA2 has a number of fault locating facilities:

- Alarm display on monitor
  (For full procedure see Description of alarms, Section 12, or alarm description issued by the machine manufacturer)
- Diagnosis, axis and spindle service data (see Section 10)
- PLC status
  (See Section 7 for use of PLC Status)
- EPROM CHECK
- Watchdog (red LED) on OP CPU
- Watchdog (red LED) on COM and NC CPU (on EPROM submodule)
- Watchdog (red LED) on Servo CPU (on EPROM submodule)
- Red LED and green LED on PLC CPU 135 WB
- Green LED at NC power supply unit
  (5V test sockets at NC and operator panel power supply units)

Three states can be indicated by the watchdogs (red LEDs) on the individual CPU modules:

- LED off: CPU cycle operation
- LED on: CPU STOP
- LED flashing
  The EPROM check has responded (does not apply to PLC CPU).

See Section 5.2 for notes on CPU monitoring:

After replacing modules and software, the PLC must be switched to the Stop status until the installation routines have been completed in overall reset mode.

Each Power On outputs the startup diagnostics of the SINUMERIK 880 GA2 on the operator panel.
When starting up for the first time and after an operator panel battery failure, an operator panel Power On routine must be executed with switch position 8 (on the OP CPU) (see Section 6.1.1).

13.2 Faults in operator panel area

The watchdog (red LED) on the operator panel CPU indicates whether the CPU is engaged in proper cyclic operation.

Each Power On outputs the startup diagnostics of the SINUMERIK 880 GA2 on the screen.

See Section 5.2.3 for further details.

13.3 Table of faults

Status				Cause	Remedy
Alarm message on screen	LED on COM CPU	LED on OP CPU	4 LEDs on op. panel flash		
Yes, English	Lit	Clocks 1 : 1	No	EPROM check in operator panel area cross check sum error	Order new system EPROM submodule
Yes, English	Lit	Lit	No	EPROM check in operator panel area submodule incorrectly located	Insert submodule in correct location
No	Lit	Lit	Yes	Fault on MPC link after "POWER ON"	a) "POWER ON" b) Check opt.-fib. conductor c) Replace COM CPU d) Replace OP CPU
Yes/No	Unlit	Unlit	Yes	Fault on MPC link in active operation or COM CPU failure	a) "POWER ON" b) Check opt.-fib.conductor c) Check opt.-fib. conductor for permissible length d) Replace COM CPU e) Replace OP CPU
No, with display refresh	Unlit	Lit	No	OP CPU failure	a) Check bootstrapper b) Replace OP CPU
Yes	Unlit	Unlit	No	NC CPU failure	a) "POWER ON" b) Check bootstrapper c) Check NC MD d) Check jumperings on: - NC CPU - measuring circuits - memory module - bus PCB e) Replace NC CPU

Status				Cause	Remedy
Alarm message on screen	LED on COM CPU	LED on OP CPU	4 LEDs on op. panel flash		
No	Clocks 1 : 1	Lit	Yes	EPROM check on COM CPU	Check system modules
No	Unlit	Lit	Yes	Voltages in central controller not OK (LED on NC power supply unit not lit)	a) 230 V available? b) NC ON operative? c) Withdraw NC power supply unit and operate as a stand-alone unit d) Replace NC power supply unit
No	Lit	Unlit	No	Voltages in operator panel power supply unit not OK	a) 230 V available? b) NC ON operative? c) Replace op. panel power supply unit
No, no display	Unlit	Unlit	No	Data transfer to screen defective or system not switched on	System switched on? a) Check cable between video module and monitor b) Check jumperings on video module c) Check brightness and contrast d) Replace video module e) Replace monitor
No, no display	Lit	Lit	Yes	• Dual/Multiport RAM defective	

13.4 Module exchange

After removing or exchanging modules you must perform the following actions.

SINUMERIK 880 GA2

Module or location		6FX 1...	Clear/ load NC/ PLC MD	Softkey user memory format	Softkey clear part program memory	Com- plete installa- tion	PLC general reset	PLC cold restart	Further com- ments
Multiport dual-tier	Variante 5, 7	136-8 BA01	X	X		X	X	X	
Dualport single-tier	Variante 1, 2, 3	124-0 BA03	X	X		X	X	X	
Power supply central controller		6EW 1861-2A.							
Power supply operator panel		6EW 1861-3A.							
Exchange backup battery while	Control switched on	/							
	Control switched off	/	X	X	X	X	X	X	
COM - CPU		147-4 BB00		X	X				
EPROM submodule for COM CPU		124-1 CC00		X	X				
INT-MEM submodule		147-5 BA01		X	X				
NC CPU		147-4 BB00		X					
EPROM submodule for NC CPU		124-1 CC00							
Servo CPU		136-3 BB01							
EPROM submodule for servo CPU		124-1 CB00							
PLC CPU		138-6 BL01					X	X	
UMS [1)]									

Further remarks:

The UM submodule can only be exchanged in installation mode (rotary switch of COM CPU in position "1").

Caution:

Modules and submodules must only be removed or exchanged when not under a voltage. EEC protection regulations must be observed.

Siemens AG AUT V250 P. O Box 48 48 W-8500 Nuremberg 1 Federal Republic of Germany	**Suggestions** **Corrections** For Publication/Manual: SINUMERIK 880 GA2 Software Version 1 Installation Guide Service Documentation
From: Name Company/Dept. Address: Telephone: /	Installation Instructions Order No.: 6ZB5 410-0HG02-0AA0 Edition: 01.93 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 48 48, W-8500 Nuremberg 1
Federal Republic of Germany

Siemens Aktiengesellschaft

Order No.: 6ZB5 410-0HG02-0AA0
Printed in the Fed. Rep. of Germany

Progress
in Automation.
Siemens