12 Alarms

12.1 General

The control contains permanently active monitors which detect malfunctions in the NC, interface controller and machine at such an early stage that damage to the workpiece, tool or machine is largely ruled out.

In the event of a malfunction, machining is first interrupted and the drives shut down, the cause of the fault being stored and displayed as an alarm. At the same time, the PLC is informed that an NC alarm is present.

Monitors exist for the following:

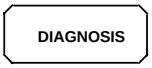
- Read-in
- Format
- Measuring circuit cables
- Position encoder and drive
- Contour
- Spindle speed
- Enable signals
- Voltage
- Temperature (DMP submodules only)
- Microprocessor
- Serial interfaces
- Data transfer between NC and PLC
- Condition of back-up battery
- System program memory
- User program memory

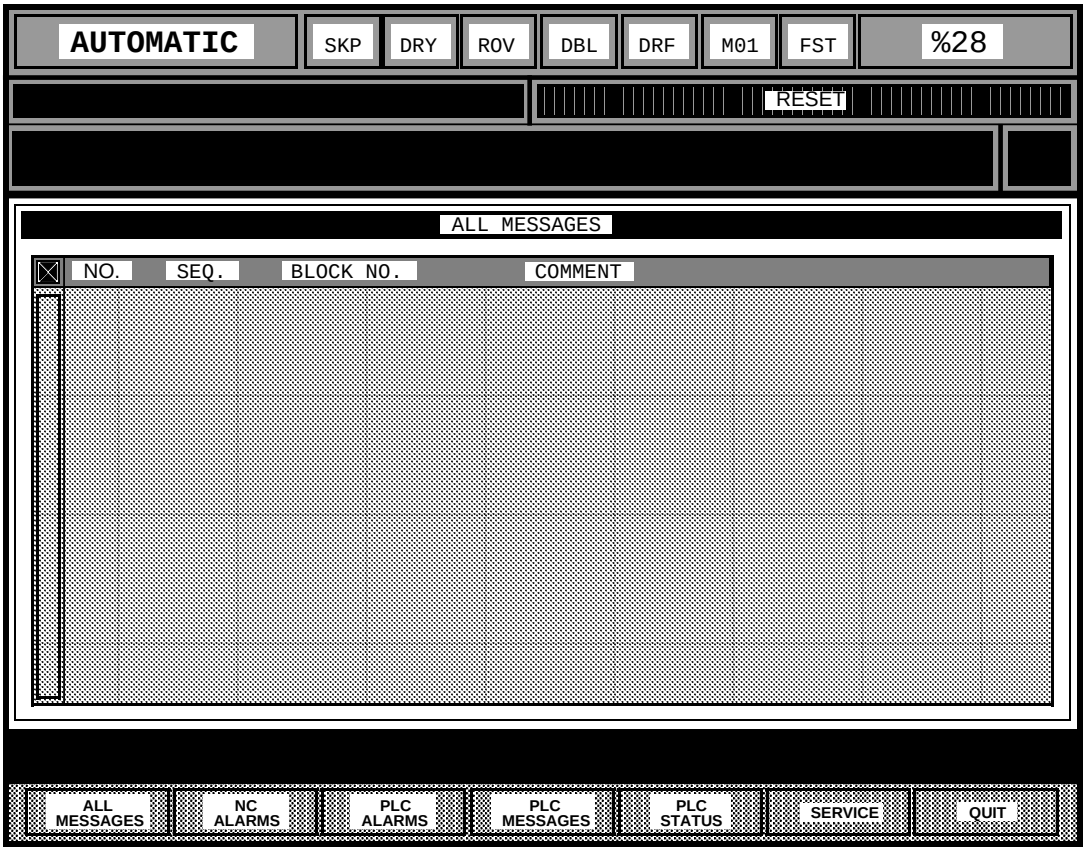
The PLC user also has an additional 64 PLC alarms and 64 PLC messages at his disposal. The associated texts are transferred to the buffered storage area of the control via the V.24 (RS232 C) interface.

12.2 Display of alarms and messages

In the event of a fault, the NC can identify and display several different alarms and messages at the same time.
However, only the lowest number with associated text is displayed in the alarm and message field. On the SINUMERIK 805, a list of all alarms and messages can be called as follows:

1. Operator area key:  Branches from any machine area menu to the basic menu of the data area.

2. Softkey: 



- The block number indicates the NC part program block in which the fault occurred.
- The ordinal number (SEQ.) indicates the sequence in which the alarms occurred. The ordinal numbers range from 1 to 99. The counter is reset when the control is switched on (POWER ON).
- The number (NO.) indicates the alarm number.
- The special alarm groups (NC alarms, PLC alarms, PLC messages) can be called by pressing the appropriate softkey.

12.3 Overview of alarms and messages / clear mode

Type	Numbers	Groups		Clear as follows:	
NC alarms	1...15 40...99 132* 136* 1)	Power On alarms		Switch control off and on	POWER ON
	16 : 39	V.24 (RS232 C) alarms		1. Select data area 2. Press DATA TRANSFER softkey 3. Press STOP softkey	STOP
	100* : 196* 1)	Axis-specific	Reset alarms	Press RESET key on operator keyboard	
	2000 : 2999	Gen-eral			
	3000 : 3999	Acknowledge alarms		Press QUIT softkey	QUIT
	4000 : 4999	Standard Cycle alarms			
	5000 : 5099	User Cycle alarms			
PLC alarms	6000 : 6063	User alarms		Press QUIT softkey	QUIT
	6100 : 6166	PLC system and programming alarms		Switch control off and on	POWER ON
PLC messages	7000 : 7063	User messages		No acknowledgement/clearing necessary	

1): The * stands for:
"0" for alarms in axis 1
"1" for alarms in axis 2
"2" for alarms in axis 3
"3" for alarms in axis 4

12.4 Alarm list

1	**Battery alarm! Do not switch off! Change battery!**	POWER ON
Cause:	Voltage of back-up battery too low	
Scan:	– At POWER ON – Cyclically	
Explanation:	Renew battery (see Section 3.1.6, Installation Instructions). The battery voltage has dropped to such a level that buffering of the user memories is guaranteed only for a short remaining period. If this alarm is output while the control is running, it can be acknowledged with the RESET key when the battery has been changed.	
Remedy:	Renew battery (see Instruction Manual). Dispose of old batteries as special waste.	
Note:	***Do not switch off*** control before changing the battery otherwise risk of data loss!	

2	**Operator panel error**	POWER ON
Cause:	Operator keyboard not ready, cable break	
Scan:	Cyclically	

3	**PLC stop**	POWER ON
Cause:	PLC not ready; PLC components defective (DMP module)	
Scan:	Cyclically	
Effect:	– NC START disabled – Setpoint relays drop out (Setpoint 0) – NC Ready 2 cancelled – Servo enable cancelled after time in MD 156 has expired (servo enable relays drop out) – NC/ PLC interface rendered inactive – Resetting of all PLC outputs	
Explanation:	Cyclical and interrupt-driven operation of the PLC is interrupted. Travel with the machine is not possible.	
Remedy:	No block list is created on start-up with hardware faults. Read out the cause of the interrupt (I STACK) with the PG (programmer).	

4	**Unit system not allowed**	POWER ON
Scan:	– With POWER ON – After modifying MD	
Effect:	Conversion factor assumed to be 1	
Explanation:	An impermissible combination of measuring system units (position control resolution) and input system units (conversion factor greater than 10) has been selected in MD 5002.	
Remedy:	Correct MD bit 5002 and switch control off and on.	

7	**EPROM error**	POWER ON
Scan:	Cyclically	
Effect:	– NC START disabled – Setpoint relays drop out (setpoint 0) – NC Ready 2 cancelled – Servo enable cancelled after time in MD 156 has expired (servo enable relays drop out)	
Explanation:	An error was detected when verifying the check sums	
Remedy:	Consult the service organization	

8	**Wrong axes or spindle**	POWER ON
Scan:	After MD modification	
Effect:	– NC START disabled – Setpoint relay drops out (setpoint 0) – NC Ready 2 cancelled	
Explanation:	Illegal assignment entered in MD 200* or MD 400*. ***Correct:*** (0000) Axis or spindle not available at machine (permissible only with NC-MD bit 564*.7="0" (for axes)) 0100 0200 0300 0400 0500	
Remedy:	See Section 8 (MD description for MD 200* and MD 400*)	

9	**NC control signals invalid**	POWER ON
Scan:	At POWER ON	
Effect:	– NC START disabled – NC Ready 2 cancelled	
Explanation	The input limits for NC MD 311-318 (cam signals), NC MD 330-345 (standard motors) or NC MD 350-381 (rapid M functions) were violated or values entered incompatible with one another (e.g. double assignment by standard motors and cam signals).	
Remedy:	Check the following MD: – NC MD 311-318 (cam signals) – NC MD 330-345 (standard motors) – NC MD 350-381 (rapid M functions)	

10	L2 module has broken down	POWER ON
Cause:	No response from L2 submodule	
Effect:	Data traffic from/to L2 submodule no longer possible	
Remedy:	POWER ON reset	

11	Option standard motor does not exist	POWER ON
Scan:	On POWER ON	
Effect:	The set bit NC MD 564*.4 (standard motor) is ignored. The axis is not being run as a standard motor but as a normal NC axis.	
Explanation:	"Connection of standard motors" option missing	
Remedy:	– Retrofit the option – Check NC MD 564*.4 (standard motor)	

12	It's not permitted to connect both 1st and 2nd handwheels	POWER ON
Scan:	At POWER ON	
Effect:	NC-PLC interface signal NC Ready 2 is cancelled	
Explanation:	An impermissible connection combination of handwheels was entered.	
Remedy:	Connection combination according to description of function (Installation Instructions Section 11).	

13	RAM error	POWER ON
Scan:	At POWER ON	
Effect:	Error in RAM area of module	
Remedy:	– Format user memory and part program in initialization mode – Replace central controller	

16	Parity error (RS232)	STOP
Effect:	– V.24 (RS232 C) transmission interrupted – Last block declared invalid	
Explanation:	The alarm can be activated only if parity is selected. The parity of the started character (8 data bit and 1 parity bit) is incorrect.The alarm has nothing to do with the V.24 (RS232 C) character parity error for ISO or EIA tape (Alarm 23)	
Remedy:	– Check parameters of V.24 (RS232 C) interface – Test external device	

17	Overflow error (RS232)	STOP
Effect:	– V.24 (RS232 C) transmission interrupted – Last block declared invalid.	
Explanation:	The external device has transmitted a new character although the NC has not yet processed the old character	
Remedy:	– Check parameters of V.24 (RS232 C) interface – Test external device	

18	Frame error (RS232)	STOP
Effect:	– V.24 (RS232 C) transmission interrupted – Last block declared invalid	
Explanation:	– The number of stop bits is incorrect – Wrong baud rate – Number of data bits is wrong	
Remedy:	– Check parameters of V.24 (RS232 C) interface – Test external device – Number of data bits: 7 data +1 parity (set from external device)	

19	External I/O device not ready (RS232)	STOP
Effect:	No files are read in	
Explanation:	Low-level DSR signal from external device	
Remedy:	– Activate external device – Do not use DSR	

22	Time monitoring (RS232)	STOP
Explanation:	– The NC cannot output a character for 60 seconds -- external device blocks CTS (clear to send) signal for more than 60 s -- when control signals (DC1-DC4) are used, no DC1 transmission by external device for 60 s – The NC has not received a character for 60 seconds.	
Remedy:	– Check external device and switch on – Check and insert cable – Switch off time watchdog in parameter menu	

23	Character parity error (RS232)	STOP
Cause:	Tape dirty or damaged	
Effect:	– V.24 (RS232 C) transmission interrupted – Last block declared invalid	
Explanation:	Depending on the definition of program start "%" or "EOR", the NC automatically specifies ISO or EIA code, and thus the character parity, after this character has been received. When the subsequent characters were checked, it was found that one character did **not** have the specified parity.	
Remedy:	Check tape and/or data on diskette	

24	Invalid EIA character (RS232)	STOP
Effect:	– Data transmission interrupted – Last block declared invalid	
Explanation:	An EIA character with correct parity has been read in but the character is **not** defined in the EIA code.	
Remedy:	Check tape: Check EIA code for "@" and EIA code for ":"	
Note:	As the "=" character is not defined in the EIA code, the following data cannot be read in: – %TEA1 (NC machine data); – %TEA2 (PLC machine data); – %RPA (R parameters); – %TOA (tool offsets); – %ZOA (zero offsets); – Main programs and subroutines with R parameter calculations	

26	Part program block > 120 characters (RS232)	STOP
Cause:	The read-in part program block has more than 120 characters. Only the actually stored characters are counted (no blanks, no CR, ...)	
Effect:	– Data transfer interrupted – Last block not stored	
Remedy:	Split block into 2 or more blocks	

27	Data input disabled (RS232)	STOP
Cause:	"Cycle disable" interface signal is present; or NC/PLC machine data texts or L2 data read in without password input.	
Effect:	No data stored	
Remedy:	– Reset Q 78.3 via PLC status – Enter password (enquiry appears when changing machine data)	

28	Ring (circular) buffer overflow (RS232)	STOP
Effect:	– V.24 (RS232 C) transmission interrupted – Last blocks declared invalid	
Explanation:	The transmission rate is so high that the number of characters read in exceeds the number that can be processed by the NC. When the program is re-transmitted, the defective program must first be erased.	
Remedy:	– RTS signal has no effect on the input device (RTS effects input device stop) – Transmission rate (baud rate) too high	

29	Block too long (max. 254 characters) (RS232)	STOP
Cause:	The read-in block has more than 254 characters. All the read-in characters (e.g. blanks) are also counted.	
Effect:	– V.24 (RS232 C) transmission interrupted – Last block not stored	
Remedy:	Split block into 2 or more blocks.	

30	Part program memory full (RS232)	STOP
Cause:	The maximum storage area for the part program is occupied	
Effect:	– Data transmission interrupted – Last block not stored	
Remedy:	Erase programs that are not longer required and reorganize memory.	

31	No further part program input (RS232)	STOP
Cause:	The maximum number of programs specified by means of machine data has been reached.	
Remedy:	– Erase programs that are no longer required and reorganize memory – Modify MD 8 and reformat part program memory Sequence: a. "INITIALIZATION" MODE b. "Format user data" softkey c. "Format part program memory" softkey This also erases all programs.	

32	Data format error (RS232)	STOP
Cause:	– Permissible number of decades after an address is incorrect – Decimal point in wrong position – Part programs or subroutines are not correctly defined or terminated (observe header) – NC expects an "=" character, but this character is not defined in the EIA code.	
Effect:	Data transmission interrupted, last block not stored	
Remedy:	Check data to be read in	

33	Stored program differs from inputted program (RS232 C)	STOP
Cause:	Read-in and stored program not identical with same program number	
Effect:	No data stored	
Remedy:	Erase old program or rename old program	
Explanation:	If an existing program with the same program number is read in again, the two programs are compared. If they differ, Alarm 33 is triggered.	

34	Operator error (RS232) interface	STOP
Cause:	Data transmission initiated at the NC and the PLC issues a second start signal	
Effect:	No data read in	
Remedy:	Stop data input and restart	

35	**SIEMENS reader error (RS232)**	STOP
Cause:	Error message from Siemens tape reader	
Scan:	Only if the setting data for the Siemens reader are set	
Effect:	– Data transmission interrupted – Last block not stored	
Remedy:	– Restart data transmission – If error occurs again, replace Siemens reader	

104*	**D/A converter limit has been reached**	
Scan:	Cyclically	
Effect:	No direct effect. The error is included in the following error Alarm 156*	
Explanation:	The set value at the DAC is higher than input in MD 268* (max. DAC set value). No further increase in set value possible.	
Remedy:	– Operate at lower speed – Check actual values (encoder) – Check MD 268* – Check drive actuator – Check MD 364* and MD 368*	

108*	**Overflow part actual value (pulse weighting)**	
Scan:	With each axis movement (also in follow-up operation)	
Effect:	– NC START disabled – Setpoint relay drops out (setpoint 0) – NC Ready 2 cancelled – Servo enable cancelled after time in MD 156 has expired (servo enable relays drop out) – Follow-up operation – Machine actual value is lost (wrong position)	
Explanation:	The part actual value is multiplied by the control. An error resulted in register overflow with high-speed axis traversing. The reference point was then lost.	
Remedy:	– Reduce max. speed – Check MD for variable increment weighting (MD 364* and MD 368*)	

112*	**Clamping monitors (zero speed control)**	
Cause:	– Wrong position control direction – Mechanically clamped axis forced out of position – Fault on control device (actuator), tacho-generator, motor, mechanical components or NC measuring circuit hardware. – Standard motor axis has overtravelled exact stop limit coarse (NC MD 380*)	
Scan:	– At standstill – During clamping – During deceleration	
Effect:	– NC START disabled – Set value 0 – Servo enable cancelled after time in MD 156 has expired (servo enable relay drops out) – Follow-up operation	
Explanation:	During positioning, the following error could not be reduced more rapidly than the time input in MD 156. During clamping, the limit specified in MD 212* was exceeded. The standard motor cannot stop within defined exact stop limit.	
Remedy:	– MD212* (clamping tolerance) must be greater than MD204* (coarse exact stop limit). – MD156 (servo enable cutoff delay) must be large enough to ensure that the following error can be suppressed within this time. – Check exact stop limits coarse and fine for standard motor. Check standard motor brake.	

116*	**Contour monitoring**	
Scan:	During processing in automatic mode but **not**: – when accelerating – when braking – at speeds less than in MD 336* (contour speed)	
Effect:	– NC START disabled – Set value 0 – Servo enable cancelled after time in MD 156 has expired (servo enable relay drops out) – Follow-up operation	
Explanation:	Tolerance band MD 332* exceeded at a speed greater than in MD 336*. During acceleration or braking, the axis did not reach the new speed within the time specified by the K_V factor. – Increase tolerance band MD 332*	
Remedy:	– Check K_V factor – Check optimization of speed controller – Check drive actuator	

132*	Control loop hardware	POWER ON
Scan:	Cyclically	
Effect:	– NC START disabled – Setpoint relay drops out – NC Ready 2 cancelled – Servo enable cancelled after time in MD 156 has expired (servo enable relay drops out) – Follow-up operation	
Explanation:	Measuring circuit differential signals – are not in phase – are short-circuited to frame – are missing altogether	
Remedy:	– Check that measuring circuit connector has been inserted (insert measuring circuit short-circuit connector to check whether the measuring circuit module is in order) – Check differential signals with oscilloscope – Exchange encoders	

136*	Measuring system dirty	POWER ON
Scan:	Cyclically	
Effect:	NC START disabled.	
Explanation:	Processing of the active program is completed. On measuring systems with contamination signal (e.g. EXE), an error is signalled by the measuring system to the NC	
Remedy:	Check measuring system	

140*	Encoder monitoring - axis	
Scan:	Cyclically	
Effect:	NC Ready 2 cancelled	
Explanation:	The measuring circuit has detected too many changes of direction within an IPO cycle. possibly caused by interference on the actual value cable.	
Remedy:	Check the measuring cycle and cable run.	

148*	**Software limit switch plus**	
152*	**Software limit switch minus**	
Scan:	With each axis movement	
Effect:	– NC START disabled – Set value 0	
Explanation:	The alarm is active only after reference point approach. Software limit switch 1 or 2 triggered depending on PLC interface signal "2nd software limit switch active".	
Remedy:	– Departure from limit switch in reverse direction – Check MD 224*, 228*, 232*, 236*	

156*	**Speed command value too high**	
Scan:	Cyclically	
Effect:	– NC START disabled – Set value 0 – Servo enable cancelled after time in MD 156 has expired (servo enable relay drops out) – Follow-up operation	
Explanation:	– Speed output in control is higher than specified in MD 264* – Motor could not follow set speed input	
Remedy:	– Check whether value in MD 264* is greater than in MD 268* – Check drive – Check measuring system – Earthing neutral point at NC? – Check drive actuator – Check position control direction (set/actual values interchanged?)	

160*	**Drift too high**	
Scan:	Cyclically	
Effect:	– NC START disable – "+" or "–"character in the  field – No traversing movement possible	
Explanation:	The drift to be compensated by the NC has risen above approx. 500 mV	
Remedy:	– Perform drift compensation Operator input: -- Select service menu in data area -- Press DRIFT COMPENSATION ...softkey for the required axis – Check whether drift was correctly adjusted on the drive unit – Check drive actuator – Check earthing arrangement	

168*	Servo enable for trav. axis not received	
Scan:	With each axis movement	
Effect:	– NC START disabled – Set value 0 – Servo enable cancelled after time in MD 156 has expired (servo enable relay drops out) – Follow-up operation	
Explanation:	Axis-specific servo enable cancelled by PLC user program during traversing movement	
Remedy:	Check PLC program	

172*	Working area limit plus	
176*	Working area limit minus	
Scan:	During processing in automatic mode During axis traversing in JOG mode	
Effect:	– NC START disabled – Set value 0	
Explanation:	Working area limitation in setting data has been reached	
Remedy:	– Check working area limitation in setting data – Check program	
Note:	Working area limitation in JOG mode only when NC-MD 5003 bit 6 is set.	

184*	Stop behind reference point cam	
Scan:	During reference point traversing	
Effect:	– NC START disabled – Set value 0 – Reference point not reached	
Explanation:	The axis was stopped between the reference cam and the zero point of the measuring system during reference point approach.	
Remedy:	Repeat reference point approach.	

2000	Emergency Stop
Scan:	Cyclically
Effect:	– NC start disabled – Setpoint 0 – Controller enable cancelled after time in MD 156 has expired (controller enable relays drop out) – Follow-up operation
Explanation:	The Emergency Stop signal is sent from the PLC to the NC
Remedy:	– Check using PLC-STATUS whether Q 78.1=0 – Check whether Emergency Stop cam is traversed or Emergency Stop button is operated – Check the PLC program
Note:	According to statutory regulations, the Emergency Stop status must be selected both by the control (software) and by the hardware (with relays).

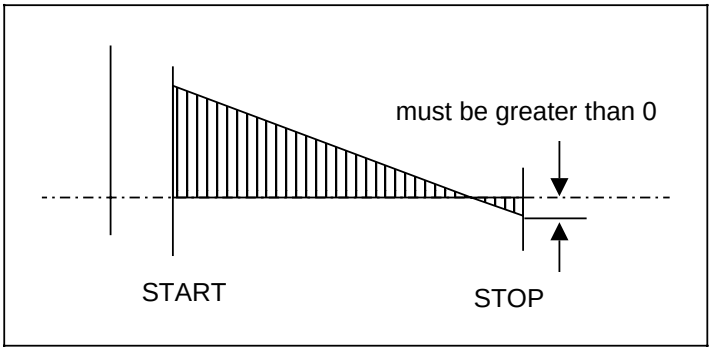
2031	Evaluation (weighting) factor too high (MD 388*)
Scan	With each axis movement
Effect:	– NC start disabled – Setpoint 0
Explanation:	– processing stop Check MD 388*

2032	Stop during threading
Effect:	– Set value 0 – NC START disabled
Explanation:	A stop has occurred in the revolutional feedrate during thread cutting, resulting in destruction of the thread.

2034	Speed reduction area
Explanation:	The software pre-limit switch has been overrun and the axes braked to reduction speed
Remedy:	– Check program – MD0: pre-limit switch – MD1: speed behind pre-limit switch (reduction speed)

2035	Programmed feed rate too high
Explanation:	The programmed speed is greater than the maximum path speed resulting from the maximum speeds of the axes
Remedy:	– Program slower path speed – Check MD 280* max. path speed – Check MD 296* when approaching reference point

2036	G35 thread lead decr. error
Scan: Significance:	During thread cutting The pitch decrease in the thread is so great that a diameter of 0 or less would result at the end of the thread.
Remedy:	Program smaller pitch decrease or shorter thread.



2037	Progr. S-value too high
Explanation: Remedy:	The programmed spindle speed "S" is greater than "12 000" Program slower spindle speed (S value limited to "12 000" in the control).

2039	Reference point not reached
Scan: Effect: Significance: Remedy:	In AUTOMATIC/MDA mode after NC START NC START disabled The reference point was not approached in all defined axes – Approach reference point in axes concerned – Set NC-MD 5004 bit 3 (NC START without reference point) ***Caution:*** No software limit switches effective. – Set NC-MD 560* bit 4. The approach to reference point can then be suppressed for one or more specific axes. ***Caution:*** No software limit switches apply in these axes.

2041	Program not in memory
Effect:	NC START disabled
Explanation:	– Preselected program not available in the memory – Subroutine called in program not available in the memory.
Remedy:	Look at "main program" and/or "subroutine" directories

2042	Parity error in memory
Scan:	During processing in automatic mode
Explanation:	One or more characters are deleted in the memory so that they can no longer be recognized (these characters are output as "?")
Effect:	NC START disabled
Remedy:	– Correct program in EDITOR or, if applicable, delete complete block and input again – With a large number of "?" the complete memory may have been erased; in this case check the battery and reformat the part program memory

2046	Block > 120 characters
Scan:	During processing in automatic mode
Effect:	NC START disabled
Explanation:	An "LF"is corrupted in the memory, producing a block of more than 120 characters
Remedy:	Insert "LF" or delete entire block

2047	Option not available
Effect:	NC START disabled
Explanation:	A program that is not contained in the control function complement has been programmed.
Remedy:	Correct program, check MD

2048	Circle endpoint error (circle centre error)
Effect:	NC START disabled
Significance:	– Programmed circle end point not on circle – End point is displaced by more than the limit input in MD 7 – No geometry in the 1st block of the contour subroutine with L95 stock removal cycle
Remedy:	Correct program

2057	Option thread/feedrate per rev. not available
Significance:	– A thread has been programmed with G33, G34, G35 although this function is **not** implemented in the control. – Revolutional feedrate G95 has been programmed
Remedy:	– Correct program – Check MD

2058	Option 3D interpolation not available
Explanation:	– 3 axes programmed simultaneously – Programmed block results in movement of 3 axes
Remedy:	– Correct program – Check MD

2059	Program error G92
Explanation:	– Use of an illegal address character – G92 programmed with address "P"
Remedy:	G92 is allowed only with address "S" (programmed spindle speed limitation)

2060	TO, ZO program error
Significance:	– Unavailable tool offset number selected – Selected zero offset or tool offset value too large – Type (P1) of called tool offset defined with 0

2061	General program error
Effect:	NC START disabled
Remedy:	– Select "Current block display" image and check block **after** the active block.

2062	Feed missing / not programmed	
Cause:	– No F value programmed – F value too small (machine data) – Programmed revolutional feedrate G95 too great – No revolutional feedrate programmed. – An axis was programmed as a simultaneous axis. However, no simultaneous feed applies for this axis at present.	
Remedy:	Program feedrate correctly	

2063	Thread lead too high	
Effect:	NC START disabled	
Explanation:	Thread pitch of more than 400 mm/rev (16 inches/rev) programmed	
Remedy:	– Program smaller thread pitch – Possibly run program on a machine with SINUMERIK 850 (max. speed 2000 mm/rev)	

2064	Rounding error for rotary axis	
Scan:	During processing in AUTOMATIC/MDA mode	
Explanation:	In the case of rounding to half or full degrees with a rotary axis, the control monitors the programmed positions to ensure that rounding has been correctly performed.	
Effect:	– NC START disabled – Programmed path in block is **not** traversed	
Remedy:	– Program correct position in rotary axis – Check MD 560* bits 2 and 3	
Note:	In the JOG and JOG-INC modes, the control automatically rounds to valid values; in the AUTOMATIC and MDA modes, the control monitors only the programmed positions and does not round automatically.	

2065	Programmed position behind sw limit switch	
Scan:	During processing in AUTOMATIC/MDA mode	
Effect:	– NC START disabled – Programmed path is **not** traversed	
Explanation:	The programmed end position of the block is behind the software limit switch.	
Remedy:	– Check program – Check MD224*, 228*, 232*, 236* depending on PLC interface signal "2nd software limit switch active"	

2066	**Thread lead increase/decrease too high**
Scan:	During processing in AUTOMATIC/MDA
Effect:	NC START disabled
Explanation:	Thread pitch increase or decrease of more than 16 mm/rev (0.6 inch/rev) programmed
Remedy:	Program smaller thread pitch increase/decrease

2067	**Max. speed of an axis = 0**
Scan:	During processing in AUTOMATIC / MDA
Effect:	NC START disabled
Explanation:	The maximum speed of an axis programmed in the block is ZERO.
Remedy:	Check MD 280*

2068	**Programmed position behind working area limit**
Scan:	During processing in AUTOMATIC MDA
Effect:	– NC START disabled – Programmed path is **not** traversed
Explanation:	The programmed end position of the block is behind the working area limitation in one or more axes
Remedy:	– Check working area limitation (positive and negative) – Modify working area limitation with G25/G26 in the program

2072	**Incorrect input value (blueprint prog.)**
Explanation:	Value input not calculable for contour definition calculation

2073	**No intersection point (blueprint prog.)**
Explanation:	No intersection is obtained with programmed values when calculating contour definition

2074	**Incorrect angle value (bluepring prog.)**
Explanation:	– Angle programmed greater than or equal to 360° – Angle value not practical for defined contour

2075	**Incorrect radius value (blueprint prog.)** 
Explanation:	– Radius too large – Radius not permitted with defined contour

2076	**Incorrect G02 / G03 (blueprint prog.)** 
Explanation:	– Circle direction **not** possible with defined contour

2077	**Incorrect block sequence (blueprint prog.)** 
Explanation:	Several blocks are required for calculating contour definition: – Block sequence incorrect – Data not sufficient (under-determined) ***Example:*** N10...B15 LF N20...G3 I20 LF

2078	**Incorrect input parameter (blueprint prog.)** 
Explanation:	– Programmed parameter sequence not allowed – Parameter sequence incomplete for defined contour ***Example:*** N10...X60 B15 LF (Z axis missing) N20...X90 B10 LF

2081	**Program block with TRC/CRC not allowed** 
Explanation: Remedy:	With tool nose/cutter radius compensation (G41/G42) selected, the following functions must not be programmed: G33, G34, G35, G58,G59, G92, M19 S..., – Program G40 first – Cancel with G41/G42 D00 (CRC/TNRC)

2082	CRC not determinable
Significance:	Axes of selected CRC plane do not exit
Remedy:	– Check MD 548*, 550*, 552* (basic setting of G16) – Select correct plane with G16

2152	Spindle speed too high
Scan:	Only when MD 520* bit 2 is set (encoder available)
Explanation:	The spindle speed is higher than specified in the data (see MD 4450)
Remedy:	– Program lower S value – MD 403* - 410* (max. spindle speed for gears 1 to 8) – MD 445* (tolerance band for max. spindle speed) – MD 451* (max. spindle speed) – Gear correctly selected by PLC? – G92 S... wrongly programmed for v constant

2153	Control loop error (spindle)
Scan:	Cyclically
Effect:	– NC START disabled – Setpoint relay drops out Setpoint 0 – NC Ready 2 cancelled – Spindle servo enable is cancelled after time in MD 4470 has expired
Explanation:	As for Alarm 132*
Remedy:	As for Alarm 132*

2154	Measuring system dirty (spindle)
Scan:	Cyclically
Explanation:	On measuring systems with contamination signal, the measuring system has signalled an error to the NC
Effect:	NC START disabled
Remedy:	Check measuring system

2155	Option M 19 not available
Scan:	During processing in AUTOMATIC / MDA
Effect:	NC START disabled
Explanation:	"M19 S ..." programmed in part program although this function is not available
Remedy:	– Correct program – Retrofit "M19" Option

2171	Approach not possible	
Explanation:	The control supplements max. one axis in the programmed plane. Where two axes must be supplemented in the programmed plane, no start-up is possible	
Remedy:	– Check NC program for complete axis programming in approach block – Programming deselection block immediately after selection block is not allowed (no calculable tangent).	

2172	Retract not possible	
Explanation:	See Alarm 2171	
Remedy:	– Check NC program for complete axis programming in approach block – With G48 deselection movement programmed (shutdown and start-up), a start-up movement must be programmed	

2173	Wrong approach / retract plane	
Explanation:	The select/deselect movements of the smooth approach/retract function are executed with reference to the selected plane G16, G17, G18, G19.	
Remedy:	Check whether a plane change is programmed after the selection or in the deselection block in the NC program	

3000	General program error	QUIT
Explanation:	A general non-accurate programming error has been made in a block in the program. ***Example:*** – The programmed axis is not available at the machine – Incorrect interpolation parameters programmed	
Remedy:	Check the faulty block in the part program. The number of the block is shown under block number in the "Messages" menu.	

3001	More than 5 geometry parameters	QUIT
Explanation:	More than 5 geometry parameters such as axes, interpolation parameters, radii, angles etc. have been programmed in the block	
Remedy:	As for Alarm 3000	

3002	Polar radius programming error	QUIT
Explanation:	Programming of the following omitted in block with polar/radius programming: – angle – radius – centre point coordinates	
Remedy:	As for Alarm 3000	

3003	Invalid address	QUIT
Explanation:	– The programmed address is not defined in the machine data. The axis names for basic setting plane G16 (MD 548*, 550*, 552*) do not correspond to the defined axis designations (MD 568*)	
Remedy:	– As for Alarm 3000 – Correct the machine data	

3004	CL800 error	QUIT
Explanation:	– @ function not available – Incorrect address after @ – Number of addresses after @ incorrect – Values in K, R or P not permissible – Number of decades too high – No decimal point allowed – Jump address incorrectly defined – System memory location (NC-MD, PLC-MD, TO, ...) not available – Bit number too high – Sine or cosine angle incorrectly stated	
Remedy:	– @ according to Programming Guide – Only K, R and P addresses allowed – Jump addresses: forwards with "+" backwards with " – " – Check validity of values in stated addresses – If applicable, select decoding single block (DEC-SBL) and check program again	

3005	Blueprint prog. error	QUIT
Explanation:	The coordinates in blueprint programming have been defined in such a way that no intersection is produced	
Remedy:	As for Alarm 3000	

3006	**Wrong block structure**	QUIT
Explanation:	– more than 3 M functions programmed in the block – more than 1 S function programmed in the block – more than 1 T function programmed in the block – more than 1 H function programmed in the block – more than 4 aux. functions programmed in the block – more than 3 axes with G00 / G01 programmed in the block – more than 2 axes with G02 / G03 programmed in the block – G04 programmed with addresses other than "X" or "F" – M19 programmed with addresses other than "S" – Incorrect or no interpolation parameters with G02 / G03 (MD 304*) – the special auxiliary functions for the analog output have been programmed incorrectly.	
Remedy:	As for alarm 3000	

3007	**Error in programming setting data**	QUIT
Explanation:	– G25 / G26 programmed – G92 programmed with an address other than "S" – M19 programmed with an address other than "S"	
Remedy:	As for alarm 3000	

3008	**Subroutine error (M17 missing, ...)**	QUIT
Explanation:	– Subroutine call without number of passes "P" – M30 programmed as program end – M17 missing at program end – 4th nesting depth activated (Only 3 subroutine levels possible on the SINUMERIK 805) – M17 programmed in main program	
Remedy:	As for Alarm 3000	

3009	**Error in part program/part program type**	QUIT
Explanation:	Of no significance on the SINUMERIK 805	

3010	Intersection error	QUIT
Significance:	This error can occur in conjunction with stock removal cycle L96 if: – Contour program programmed without G0, G1, G2, G3 – @ 714 programmed in contour program – Incorrect plane in contour program – No intersection found – More than quarter circle programmed in contour program – No geometry in first block of contour program of L95 stock removal cycle	
Remedy:	As for Alarm 3000	

3011	Too many axes programmed/axes programmed twice	QUIT
Explanation:	– An axis has been programmed twice in the same block – More axes have been programmed than are available on the machine	
Remedy:	As for Alarm 3000	

3012	End block in the memory not available	QUIT
Explanation:	– Program not terminated with M02 / M30 / M17. – Block number stated in the jump (@ 100, 11x, 12x, 13x) was not found in the specified direction. – On block search with calculation, the sought block number is not in the program.	
Remedy:	As for Alarm 3000	

3015	Rotary axis not allowed	QUIT
Explanation:	– A rotary axis was programmed as a simultaneous axis although that is not permitted. – A standard motor axis was defined as a rotary axis.	
Remedy:	– Change NC program so that the rotary axis is no longer addressed as a simultaneous axis. – Deselect rotary axis definition	

3016	External data input error	QUIT
Effect:	Data transfer interrupted	
Explanation:	With external data input from the PLC to the NC: – Code incorrect – Value too great – Dimensional identifier impermissible – Option not available	
Remedy:	– Check PLC program – Check NC MD, PLC MD	

3018	Distance from contour too large	QUIT
Scan:	After NC Start (AUTOMATIC)	
Effect:	- Machining stop	
Explanation:	After returning to the circle, the distance to the circle contour is too great (MD 9)	
Remedy:	Observe MD 9, approach closer to the contour	

3019	Option 2nd V.24 not available	QUIT
Explanation:	The 2nd V.24 (RS 232 C) interface has been activated by the PLC or softkey without the Option being available	
Remedy:	– Implement data transfer via the 1st V.24 (RS 232 C) interface – Retrofit Option C62 (2nd V.24 (RS 232 C) interface)	

3020	Option not available	QUIT
Explanation:	A function has been programmed that is not available in the control	
Remedy:	– As for Alarm 3000 – Retrofit Option	

3021	TRC/CRC contour error	QUIT
Scan:	With TNRC/CRC selected ***NOT:*** – in selection block – in deselection block	
Explanation:	Correction calculation produces a traversing movement opposing the programmed movement.	

3072	Alarm text not available	QUIT
	(available soon)	

3081	CRC not selected for approach	QUIT
Explanation:	The "Smooth approach to and exit from contour" function is possible only with cutter radius compensation effective. G41/G42 D0 is considered as selected in this case.	
Remedy:	Select CRC	

3200	L2 bus parameter not set	QUIT
Cause:	Softkey has be pressed to transfer a link into the link list although the bus parameters have not been set.	
Effect:	SINEC L2 submodule does not go to the bus	
Remedy:	Enter bus parameters completely	

3201	L2 link running	QUIT
Cause:	Softkey has been pressed to transfer changed bus parameters to the L2 submodule although a link is still working with the old bus parameters.	
Effect:	Changed bus parameters are not transferred.	
Remedy:	Switch the existing link inactive.	

3202	Overstep of value area	QUIT
Cause:	A bus parameter or link parameter has an impermissible value (value range exceeded or defined twice)	
Effect:	Changed values were not transferred into L2 submodule.	
Remedy:	Correct values	

4100	No D No. active	QUIT
Explanation:	TNRC/CRC selected without stating a D No. within a standard cycle	
Remedy:	Check part program	

4101	Tool radius = 0	QUIT
Explanation:	Cutter radius stated as 0. This leads to errors with standard cycles.	
Remedy:	Enter radius in D No.	

4102	Cutter radius too great	QUIT
Explanation:	Use of this cutter would lead to contour errors with some standard cycles	
Remedy:	Program different cutter	

4103	Tool too wide	QUIT
Explanation:	Grooving tool too wide for standard grooving cycle	

4120	No direction of spindle rotat. programmed	QUIT
Explanation:	No direction of spindle rotation programmed before calling standard cycle	
Remedy:	Modify part program	

4121	Spindle not in tolerance range	QUIT
Explanation:	If spindle speed fluctuation is too great in conjunction with standard cycle	
Remedy:	Check drive actuator MDs	

4140	Machined part diameter too small	QUIT
Explanation:	The machined part diameter was input too small when parameterizing a standard cycle	
Remedy:	Check part program	

4180	Option not available	QUIT
Explanation:	The called standard requires an Option that is not available in the control	
Remedy:	Retrofit Option	

4200	Check definition R (Nxxx)	QUIT
Explanation: Remedy:	Initialization parameter incorrectly defined Define initialization parameter correctly	

5000 ⋮ 5099	User cycle alarm	QUIT
Cause: Remedy:	Alarm called in user cycle with help of @ 4C0 Check the cycle	

6000 ⋮ 6063	PLC user alarms	QUIT
Explanation: Remedy:	Initiation bit set in PLC user program Determined by manufacturer	

6100	Signal converter missing	POWER ON
Cause: Effect: Remedy:	Load or transfer command to non-available I/Os, e.g. L PB, T PB PLC STOP Check I/O addressing and/or STEP 5 program	

6101	Illegal MC5 code	POWER ON
Cause: Effect: Remedy:	STEP 5 command cannot be interpreted PLC STOP – Check and/or reload PLC program – Analyze I STACK	

6102	Illegal MC5 parameter	POWER ON
Cause: Effect: Remedy:	Impermissible parameter type (I, Q, F, C, T) or parameter value PLC STOP – Check PLC program – Analyze I STACK	

6103	Transfer to missing DB	POWER ON
Cause:	L DW or T DW without prior "opening" (C DB ...) of a data block	
Effect:	PLC STOP	
Remedy:	Check PLC program Check I STACK	

6105	Missing MC5 block	POWER ON
Cause:	A block has been called that is not available in the control (OB, PB, SB, FB).	
Effect:	PLC STOP	
Explanation:	e.g. OB2 not available	
Remedy:	Input missing block	

6106	Missing data block	POWER ON
Cause:	A data block has been called that is not available in the control	
Effect:	PLC STOP	
Remedy:	Input missing DB.	

6107	Invalid segment LIR/ TIR	POWER ON
Cause:	LIR: Segment No. 0 to A permissible TIR: Segment No. 0 to 6 permissible.	
Effect:	PLC STOP	
Explanation:	See PLC Programming Guide	
Remedy:	Correct program.	

6108	Invalid segment TNB/TNW	POWER ON
Cause:	Source: Segment No. 0 to A permissible Target: Segment No. 0 to 6 permissible.	
Effect:	PLC STOP	
Explanation:	See PLC Programming Guide	
Remedy:	Correct program.	

6109	Overflow block stack	POWER ON
Cause:	Nesting depth greater than 12.	
Effect:	PLC STOP	
Explanation:	e.g. when a block calls itself	
Remedy:	Correct program.	

6110	Overflow interrupt stack	POWER ON
Cause:	More than two I STACK entries	
Effect:	PLC STOP	
Explanation:	Cyclic program (OB1) is interrupted by alarm program (OB2) and alarm program interrupts itself	
Remedy:	See processing delay OB2, Alarm 6162.	

6111	MC5 command STS	POWER ON
Cause:	STS command programmed in FB	
Effect:	PLC STOP	
Explanation:	Immediate termination of STEP 5 program processing	

6112	MC5 command STP	POWER ON
Cause:	STP command programmed	
Effect:	PLC STOP	
Explanation:	PLC STOP on termination of STEP 5 program processing	

6113	Illegal MC5 timer / counter	POWER ON
Cause:	STEP 5 timer or counter not available or not enabled by MD	
Effect:	PLC STOP	
Remedy:	– Correct program – Modify PLC MD 6.	

6114	Function macro	POWER ON
Cause:	Error in an assembler function block (basic program)	
Effect:	PLC STOP	
Remedy:	See Programming Guide, Function Macros.	

6115	System commands disabled	POWER ON
Cause:	Programmed command LIR, TIR, TNB, TNW.	
Effect:	PLC STOP	
Remedy:	Set PLC MD 2003 bit 4 (enable system commands)	

6116	MD 0000: Alarm byte No.	POWER ON
Cause:	PLC MD 0 set greater than 31.	
Effect:	PLC STOP	
Remedy:	Correct MD	

6117	MD 0001: CPU load	POWER ON
Cause:	PLC MD 1 greater than 20 %.	
Effect:	PLC STOP	
Remedy:	Correct MD	

6118	MD 0003: Alarm run time	POWER ON
Cause:	PLC MD 3 greater than 2500 µs.	
Effect:	PLC STOP	
Remedy:	Correct MD	

6119	MD 0005: Cycle time	POWER ON
Cause:	PLC MD 5 greater than 320 ms.	
Effect:	PLC STOP	
Remedy:	Correct MD	

6121	MD 0006: Last MC5 time	POWER ON
Cause:	PLC MD 6 greater than 31.	
Effect:	PLC STOP	
Remedy:	Correct MD	

6124	Gap in MC5 memory	POWER ON
Cause:	Discontinuous sequence of valid and invalid blocks	
Effect:	PLC STOP	
Remedy:	Overall reset and reload the PLC program	

6125	Dual assignment of inputs	POWER ON
Cause:	An input address has been used twice (overlapping of two DMP module input areas)	
Effect:	PLC STOP depending on PLC MD 2003 bit 2	
Remedy:	Check PLC MD 10-19	

6126	Dual assignment of outputs	POWER ON
Cause:	An output address has been used twice (overlapping of two DMP module output areas)	
Effect:	PLC STOP depending on PLC MD 2003 bit 2	
Remedy:	Check PLC MD 10-19	

6127	Alarm byte missing
Cause:	Selected alarm input byte not available as hardware
Effect:	PLC STOP
Remedy:	– Change PLC MD 0 – Set address decoding for alarm byte

6130	Synchro error basic program	POWER ON
Cause:	Synchronization pattern no longer correct with assembler function blocks	
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program if applicable	

6131	Synchro error MC5 program	POWER ON
Cause:	Synchronization pattern no longer correct with STEP 5 program blocks	
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program	

6132	Synchro error MC5 data	POWER ON
Cause:	Synchronization pattern no longer correct with STEP 5 data blocks	
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program	

6133	Illegal block basic program	POWER ON
Effect:	PLC STOP	
Remedy:	Exchange system software	

6134	Illegal block MC5 program	POWER ON
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program	

6135	Illegal block MC5 data	POWER ON
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program	

6136	Summing error MC5 block	POWER ON
Effect:	PLC STOP	
Remedy:	PLC GENERAL RESET, reload PLC program	

6137	Summing error basic program	POWER ON
Effect:	PLC STOP	
Remedy:	Exchange system software	

6138	No response from MPC	POWER ON
Cause:	Cable break, wrong submodule No. set or similar	
Effect:	PLC STOP depending on PLC MD 2003 bit 2	
Remedy:	Check cable and/or carrier module	

6139	MPC transfer error	POWER ON
Cause:	Same submodule No. set with 2 DMP submodules	
Effect:	PLC STOP depending on PLC MD 2003 bit 2	
Remedy:	Check submodule No.	

6140	PLC MD 10 - MD 19: DMP address start wrong	POWER ON
Cause:	Incorrect input in PLC MD 10-19	
Effect:	PLC STOP depending on PLC MD 2003 bit 2	
Remedy:	Check PLC MD 10-19	

6149	Stop, using softkey PG	POWER ON
Cause:	Stop command via PG.	
Effect:	PLC STOP	
Remedy:	• PLC Start via PG, • Power On.	

6150	Timeout: User memory	POWER ON
Effect:	PLC STOP	
Explanation:	(S5 program)	
Remedy:	Analyze error fine coding, see Programming Guide	

6152	Timeout: LIR / TIR	POWER ON
Cause:	Attempted access to addresses that are not available	
Effect:	PLC STOP	
Explanation:	See Programming Guide	
Remedy:	Check segment and offset address. Hardware available?	

6153	Timeout: TNB / TNW	POWER ON
Cause:	Faulty programming or use of TNB / TNW.	
Effect:	PLC STOP	
Explanation:	See Programming Guide	
Remedy:	– Check source and target address for admissibility. – Addresses available?	

6154	Timeout: L PB / L PW / T PB / T PW	POWER ON
Cause:	Load or transfer command to I/O device that has failed.	
Effect:	PLC STOP	
Remedy:	Check I/O device or change modules	

6155	Timeout: Substitution operation	POWER ON
Effect:	PLC STOP	
Remedy:	Check PLC program.	

6156	Timeout: Not interpretable	POWER ON
Cause:	Timeout not definable by system program	
Effect:	PLC STOP	
Remedy:	– Analyze error fine diagnosis – PLC GENERAL RESET, reload PLC program	

6157	Timeout: JU FB / JC FB	POWER ON
Cause:	In the resident function macros, attempts are made to access addresses that are not available.	
Effect:	PLC STOP	
Remedy:	Check hardware	

6158	Timeout: on I/O transfer	POWER ON
Cause:	Central I/O devices no longer respond	
Effect:	PLC STOP	
Explanation:	Central I/O devices are covered during start-up. No alarm if the + 24.V supply fails.	
Remedy:	Exchange central controller	

6159	Runtime exceeded STEP 5 program	POWER ON
Cause:	The max. running time in PLC MD 1 has been exceeded.	
Effect:	PLC STOP depending on PLC MD 2003 bit 1.	
Explanation:	Analyze diagnosis DB.	
Remedy:	– Increase MD 1 CPU load rises – Set MD 2003 bit 6 Cycle time increases – Optimize PLC program time	

6160	Runtime exceeded OB2	POWER ON
Cause:	The max. running time in PLC MD 3 has been exceeded.	
Effect:	PLC STOP, depending on PLC MD 2003 bit 0.	
Explanation:	Analyze DB diagnosis.	
Remedy:	– Increase MD 3 – Optimize OB2 time.	

6161	Cycle time exceeded	POWER ON
Cause: Effect: Explanation: Remedy:	The max. running time in PLC MD 5 has been exceeded PLC STOP ***Note:*** Bit commands have a processing time of only approx. 1µs. – Increase MD 5 – Optimize PLC program time.	

6162	Processing delay OB2	POWER ON
Cause: Effect: Explanation: Remedy:	The alarm program (OB2) has interrupted itself PLC STOP, depending on PLC MD 2003 bit 0 Analyze diagnosis DB Optimize OB2 time, i.e. reduce the active processing time of the alarm program	

6163	Time supervisor (monitoring) PLC network	POWER ON
Cause: Effect: Remedy:	After 20 PLC cycles data received from L2 submodule have still not been transferred to the PLC by a RECEIVE job. Display dependent on PLC MD 2002 bit 2 Check the PLC program, especially the RECEIVE job in question.	

6164	DMP protection function output (short-circuit)	POWER ON
Cause: Remedy:	Short-circuit at a DMP module output Analyze diagnosis DB1	

6165	DMP 24V supply for logic not O.K.	POWER ON
Cause: Remedy:	Failure of 24V supply or voltage below other threshold (15V) Check power supply	

6166	DMP overtemperature (> 63 deg. C)	POWER ON
Cause: Remedy:	Overtemperature in DMP module Analyze diagnosis DB1	

7000 ⋮ 7063	PLC user messages
Cause:	Initiation bit set in PLC user program