Diagnostics Guide 11/2002 Edition

sinumerik

SINUMERIK 840D/840Di/810D

SIEMENS

SIEMENS

SINUMERIK 840D/840Di/810D

Diagnostics Guide

Valid for

Control	*Software version*
SINUMERIK 840D	6
SINUMERIK 840DE (Export version)	6
SINUMERIK 840D powerline	6
SINUMERIK 840DE powerline	6
SINUMERIK 840Di	2
SINUMERIK 840DiE (Export version)	2
SINUMERIK 810D	3
SINUMERIK 810DE (Export version)	3
SINUMERIK 810D powerline	6
SINUMERIK 810DE powerline	6

Edition: 11.02

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 the "Remarks" column:

A .... New documentation.

B .... Unrevised reprint with new order no.

C .... Revised edition with new status.
If factual changes have been made on a page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
02.95	6FC5 298-2AA20-0BP0	**A**
04.95	6FC5 298-2AA20-0BP1	**C**
09.95	6FC5 298-3AA01-0BP0	**Description of differences**
03.96	6FC5 298-3AA20-0BP0	**C**
08.97	6FC5 298-4AA20-0BP0	**C**
12.97	6FC5 298-4AA20-0BP1	**C**
12.98	6FC5 298-5AA20-0BP0	**C**
08.99	6FC5 298-5AA20-0BP1	**C**
04.00	6FC5 298-5AA20-0BP2	**C**
10.00	6FC5 298-6AA20-0BP0	**C**
09.01	6FC5 298-6AA20-0BP1	**C**
02.02	6FC5 298-6AA20-0BP2	**C**
11.02	6FC5 298-6AA20-0BP3	**C**

This manual is included in the documentation on CD-ROM (**DOCONCD**)

Version	Order No.	Remarks
11.02	6FC5 298-6CA00-0BG3	C

Trademarks
SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are trademarks of Siemens. Other product names used in this documentation may be trademarks which, if used by third parties, could infringe the rights of their owners.

Further information is available on the Internet under:
http://www.ad.sinumerik.de/sinumerik

This publication was produced with FrameMaker V7.0 and Designer V7.0.
Reproduction of this document and/or transmittal thereof to third parties, as well as utilization or disclosure of the contents thereof, are not permitted unless express authorization is given in writing. Offenders are liable to the payment of damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG, 1995–2002. All rights reserved

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

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

Subject to change without prior notice

Order No. **6FC5 298-6AA20-0BP3**
Printed in Germany

Siemens Aktiengesellschaft

Preface

Structure of the documentation

The SINUMERIK documentation is divided up into three levels:

- General documentation
- User documentation
- Manufacturer/Service documentation.

This manual is intended as a work of reference. It allows the operator at the machine tool:

- To correctly assess special situations when operating the machine.
- To ascertain the reaction of the system to the special situation.
- To utilize the possibilities for continued operation following the special situation.
- To follow references to other documentation containing further details.

Scope

This manual describes the alarms from the NC kernel (NCK) area, the PLC and the SIMODRIVE 611D drive.
Other alarms can occur from the MMC (Man-Machine Communication) areas. They are displayed on the operator panel in the form of self-explanatory text. They are documented in the section on MMC messages.
For special situations in conjunction with the integrated PLC, please refer to the SIMATIC S7-300 documentation.

Please contact your local SIEMENS office for more detailed information about documentation on SINUMERIK 840D/810D and publications which apply to all SINUMERIK controls.

Target group

This documentation is intended for manufacturers of machine tools with SINUMERIK 840D or SINUMERIK 810D and SIMODRIVE 611D.

Hotline

If you have any questions, please get in touch with our hotline:

A&D Technical Support

Tel.: ++49-(0)180-5050-222
Fax: ++49-(0)180-5050-223
e-mail: adsupport@siemens.com

If you have any questions about the documentation (suggestions, corrections), please fax or e-mail us at the following addresses:

Fax: ++49-(0)9131-98-2176
email: motioncontrol.docu@erlf.siemens.de

Fax form: See the reply form at the end of the document.

Internet address

http://www.ad.siemens.de/sinumerik

© Siemens AG, 2002. All rights reserved

SINUMERIK 840D powerline

From 09.2001

- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline

will be available with improved performance. The following hardware description contains a list of the available **powerline** modules:

References: /PHD/ SINUMERIK 840D Configuration Manual

SINUMERIK 810D powerline

From 12.2001

- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline

will be available with improved performance. The following hardware description contains a list of the available **powerline** modules:

References: /PHC/ SINUMERIK 810D Configuration Manual

Information about software versions

The software versions specified in the documentation are valid for the SINUMERIK 840D and SINUMERIK 810D controls. The software versions are related to one another; see the table:

SINUMERIK 840D		**SINUMERIK 810D**	**SINUMERIK 810D powerline**	**SINUMERIK 840i**
6.3 (09.01)	corresponds to	-	6.1 (12.01)	2.1 (07.01)
5.3 (04.00)	corresponds to	3.3 (04.00)	-	1.1 (07.00)
3.7 (03.97)	corresponds to	1.7 (03.97)	-	-

Export version

The following functions are not included in the export version:

Function	**810E**	**840DE**
Machining package five axes	-	-
Handling transformation package (5 axes)	-	-
Multiple-axes interpolation (> 4 axes)	-	-
Helical interpolation 2D+6	-	-
Synchronous actions step 2	-	O1)
Measurement level 2	-	O1)
Adaptive control	-	O1)
Continuous dressing	-	O1)
Using the compile cycles (OEM)	-	-
Multidimensional sag compensation	-	O1)

Sorting

The alarms are sorted by ascending alarm number in each section. There are gaps in the sequence.

© Siemens AG, 2002. All rights reserved

Structure of the alarm descriptions

Each alarm consists of an alarm number and alarm text. There are four description categories:

- Explanation
- Reaction
- Remedy
- Program continuation.

For a more detailed explanation of the "Reaction" category, please refer to the section: "System reactions on alarms"

For a more detailed explanation of the "Program continuation" category, please refer to the section: "Clear criteria for alarms"

NCK alarms

Table 1_1 Alarm number ranges

000 000 - 009 999	General alarms
010 000 - 019 999	Channel alarms
020 000 - 029 999	Axis/spindle alarms
030 000 - 099 999	Functional alarms
060 000 - 064 999	Cycle alarms SIEMENS
065 000 - 069 999	Cycle alarms user
070 000 - 079 999	Compile cycles, manufacturer and OEM

HMI alarms/messages

Table 1_2 Alarm number ranges, continued

100000 - 100999	Basic system
101000 - 101999	Diagnostics
102000 - 102999	Services
103000 - 103999	Machine
104000 - 104999	Parameter
105000 - 105999	Programming
106000 - 106999	Spare
107000 - 107999	OEM
109000 - 109999	Distributed systems (M to N)
110000 - 110999	HMI Embedded messages
120000 - 120999	HMI Advanced messages

611D alarms

Table 1_3 Alarm number ranges, continued

300000 - 399999	Drive

© Siemens AG, 2002. All rights reserved

PLC alarms/messages

Table 1_4 Alarm number ranges, continued

400000 - 499999	General alarms
500000 - 599999	Channel alarms[2)]
600000 - 699999	Axis/spindle alarms[2)]
700000 - 799999	User area[2)]
800000 - 899999	Sequencers/graphs[2)]
(810001 - 810009	System error messages from PLC[1)])

[1)] More detailed information is available via the diagnostics function (diagnostics buffer) of SIMATIC STEP 7.

[2)] The PLC alarms in the range 500000 - 899999 are configured and described by the machine manufacturer.

Action list

The actions described in the alarm texts ("Action %---") are explained in detail in the table in the "Action list" section.

Search helps

For better orientation, you are provided with a table of contents as well as the appendices:

- Abbreviations
- List of References.

Safety

Danger
Please check the situation at the system very carefully on the basis of the description of the alarms that have occurred. Eliminate the causes for the occurrence of the alarms and acknowledge in the manner indicated. Otherwise, the machine, workpiece, stored settings and possibly your health are at risk.

Explanation of symbols

!

Important
This notice indicates that important facts have to be taken into consideration.

Ordering data supplement

This symbol appears in this documentation whenever a described function is not contained in the standard scope of supply and has to be ordered as an option.

© Siemens AG, 2002. All rights reserved

Danger and warning concept

The following warning notices with varying degrees of significance are used in the document:

Danger
Indicates an imminently hazardous situation which, if not avoided, **will** result in death or serious injury or in substantial property damage.

Warning
Indicates a potentially hazardous situation which, if not avoided, **could** result in death or serious injury or in substantial property damage.

Caution
Used with the safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in minor or moderate injury or in property damage.

Caution
Used without safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in property damage.

Notice
Used without the safety alert symbol indicates a potential situation which, if not avoided, **may** result in an undesirable result or state.

© Siemens AG, 2002. All rights reserved

© Siemens AG, 2002. All rights reserved

Contents

1) Further information is available via the diagnostic function (diagnostic buffer) of SIMATIC STEP 7.

© Siemens AG, 2002. All rights reserved

© Siemens AG, 2002. All rights reserved

Alarms 1

1.1 Overview of system error alarms

System errors

The following alarms are system errors:

1000	1005	1013	1017
1001	1010	1014	1018
1002	1011	1015	1019
1003	1012	1016	1160

These system error alarms are not described in detail. If such a system error occurs, please note
- the alarm number,
- the alarm text and
- and the internal system error number

and send these details to

SIEMENS AG, A&D MC, System Support
Hotline
Phone: (0)180-5050-222 (Germany)
Fax: (0)9131-98-2176

Phone: ++49-(0)180-5050-222 (International)
Fax: ++49-(0)9131-98-2176

1.2 Overview of alarms

© Siemens AG, 2002. All rights reserved

1000 System error %1

Parameters: %1 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Make a note of the error text and contact Siemens A&D MC, Hotline
- Phone: 0180-5050-222 (Germany)
- Fax: 0180-5050-223
- Phone: ++49-(0)180-5050-222 (International)
- Fax: ++49-(0)180-5050-223
- email techsupport@ad.siemens.de

Program Continuation: Switch control OFF - ON.

1001 System error %1

Parameters: %1 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

1002 System error %1

Parameters: %1 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions: - Alarm display.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the Delete key or NC START.

1003 Alarm pointer for this self-clearing alarm %1 is zero

Parameters: %1 = Incorrect alarm number

Definitions: The address (zero pointer) used by the compile cycle manufacturer or by the operating system for self-clearing alarms is not allowed in the system.

Reactions: - Alarm display.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Check setCCAlarm/setAlarm (...) call.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

1004 Alarm reaction to NCK alarm incorrectly configured

Parameters: %1 = Incorrect alarm number

Definitions: The alarm reaction configured by the operating system or the compile cycles manufacturer is incorrect.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Change alarm reaction

Program Continuation: Switch control OFF - ON.

1005 Operating system error %1 parameter %2 %3 %4

Parameters:
%1 = Operating system error number
%2 = Operating system error parameter 1
%3 = Operating system error parameter 2
%4 = Operating system error parameter 3

Definitions: This alarm indicates that the operating system has detected a serious error.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Switch control OFF - ON.

1010 Channel %1 system error %2 action %3

Parameters:
%1 = Channel number
%2 = System error number
%3 = Action number/action name

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Interpreter stop

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Switch control OFF - ON.

1011 Channel %1 %3 %4 system error %2

Parameters:
%1 = Channel number
%2 = System error number
%3 = Optional parameter: Block number, label
%4 = Optional parameter: Action number, ...

© Siemens AG, 2002. All rights reserved

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key. Restart part program

1012 Channel %1 system error %2 %3 %4

Parameters:
%1 = Channel number
%2 = System error number
%3 = Parameter1
%4 = Parameter2

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the Delete key or NC START.

1013 Channel %1 system error %2

Parameters:
%1 = Channel number
%2 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key. Restart part program

1014 Channel %1 system error %2

Parameters:
%1 = Channel number
%2 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

1015 Channel %1 axis %2 system error %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error. Especially with parameter %3 (system error number) = 840001 = Problem with tool management, the identification for the axis is not contained in parameter %2, but instead, further information for the diagnostics (= Status of the data management/magazine no./location no./T no.)

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Channel not ready.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key. Restart part program

1016 Channel %1 axis %2 system error %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Mode group not ready.
- Channel not ready.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

1017 Channel %1 axis %2 system error %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = System error number

Definitions: With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reactions: - Alarm display.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the Delete key or NC START.

1018 Floating point arithmetic error in channel %1 task %2 station %3 FPU state %4 %4

Parameters: %1 = Channel number
%2 = Task ID
%3 = Station priority
%4 = FPU status

Definitions: The floating point unit of the processor has found a computational error.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- Alarm reaction delay is canceled.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

1019 Floating point arithmetic error at address %3 in channel %1 task %2 FPU state %4

Parameters: %1 = Channel number
%2 = Task ID
%3 = Code address of operation that triggered the error
%4 = FPU status

Definitions: The floating point unit of the processor has triggered an exception on account of a computational error.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Alarm reaction delay is canceled.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

1030 System error in link module error code %1 error type %2

Parameters: %1 = Hex-Zahl Link-Error
%2 = Hex-Zahl Link-Error-Type

Definitions: This alarm is not a user error. An internal error has occurred in the software of the link module. Two parameters are output with this error for debugging purposes. They provide information about the cause and location of the error.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Switch control OFF - ON.

1031 Link module generated an unspecified error %1 NCU %2 %3 %4

Parameters: %1 = Hex-Zahl unspecified status in stateOfLinkModules
%2 = NCU number
%3 = Command from link module to NCK
%4 = Status of own link

© Siemens AG, 2002. All rights reserved

Definitions: This alarm is not a user error.
- 1. If NCU== 0 -> A parameter not equal to zero was not found
- 2. If NCU not equal to zero -> An error which the NC was not able to interpret in the connection to this NCU. The error is output as a number. It is possible that the NCU link module is running a newer software version than the NC.

The other parameters are used for error localization in the NC/LINK-MODUL software.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Switch control OFF - ON.

1100 No valid firmware

Definitions: No memory card or memory card without valid firmware (license) inserted.

Reactions: - Alarm display.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000)

Program Continuation: Switch control OFF - ON.

1160 Assertion failed in %1: %2

Parameters: %1 = String (path with program name)

%2 = String (line number)

Definitions: This alarm is purely a development alarm and will not appear in a delivered software version. For an OEM customer, this alarm could indicate that an alarm has occurred in the system software. Handling of the 'assertion' allows error conditions leading to this alarm to be defined in the system software during the development phase. After the development phase, this alarm output is no longer active.

Reactions: - NC not ready.
- Channel not ready.
- Alarm reaction delay is canceled.
- The NC switches to follow-up mode.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the cause of the error in the specified software component at the specified line number.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

2000 PLC sign-of-life monitoring

Definitions: The PLC must give a sign of life within a defined period of time (machine data 10100 PLC_CYCLIC_TIMEOUT). If this does not occur, the alarm is triggered.

The sign of life is a counter reading on the internal NC/PLC interface which the PLC causes to count up with the 10 ms time alarm. The NCK also tests cyclically whether the counter reading has changed.

© Siemens AG, 2002. All rights reserved

Reactions: - Local alarm reaction.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check monitoring time frame in NCK-MD 10100 PLC_CYCLIC_TIMEOUT (reference value: 100ms).

Establish the cause of the error in the PLC and eliminate it (analysis of the ISTACK. If monitoring has responded with a loop in the user program rather than with a PLC Stop, there is no ISTACK entry).

Program Continuation: Switch control OFF - ON.

2001 PLC has not started up

Definitions: The PLC must give a sign of life within a period of time defined in MD 10120 PLC_RUNNINGUP_TIMEOUT (Default setting: 1 sec.).

Reactions: - NC Start disable in this channel.
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- Local alarm reaction.
- Alarm display.
- Interface signals are set.

Remedy:
- Please inform the authorized personnel/service department. The monitoring time in MD 10120 PLC_RUNNINGUP_TIMEOUT must be checked and adapted to the first OB1 cycle.
- Establish the cause of error in the PLC (loop or stop in the user program) and eliminate.

Program Continuation: Switch control OFF - ON.

2100 NCK battery warning threshold reached

Definitions: The undervoltage monitor of the NCK battery has reached the prewarning threshold. This is at 2.7-2.9V (nominal voltage of the battery is 3.0-3.1V at 950mAh).

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. The battery must be replaced within the next 6 weeks. After this period, the voltage can drop below the alarm limit of 2.4-2.6V if the RAMs to be buffered take up a lot of current.

Program Continuation: Clear alarm with the Delete key or NC START.

2101 NCK battery alarm

Definitions: The undervoltage monitoring (2.4-2.6V) of the NCK battery has responded during cyclic operation.

Reactions: - Alarm display.

Remedy: If the NCK battery is replaced without interrupting the power supply, no data will be lost. This means that production can continue without taking any further steps. (A buffer capacitor on the NCK holds the supply voltage for at least 30 minutes and the battery can be replaced within this time even when the control is switched off).

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

2102 NCK battery alarm

Definitions: The undervoltage monitoring (2.4-2.6V) of the NCK battery was detected during system power-up.

Reactions:
- The NC switches to follow-up mode.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Pull out the battery/fan unit from the NC module and replace the battery (type: lithium battery with lead, size 1/2AA, 850mAh, min. 3.2V).

The system must then be reinitialized because it must be assumed that data has been lost in the buffered RAM during the last power-off phase as a result of insufficient supply voltage (refer to Section 2.2 in the Installation and Start-up Guide for the procedure).

The following data might have been corrupted or entirely lost:

- NC machine data
- Drive machine data
- Option data
- Setting data
- User variable
- Global subroutines
- Cycles and macros, as well as
- PLC machine data
- PLC basic program
- PLC user program, and all
- PLC user data.

User data in the NCK and PLC (e.g. tool and workpiece data) that have been altered by the manufacturing process since the last data backup must be updated manually to match the present machine status!

Program Continuation: Switch control OFF - ON.

2110 NCK temperature alarm

Definitions: The temperature sensor has reached the response threshold of 60°C 2.5°C.

Reactions: - Alarm display.

Remedy: In order to reset the sensor, the temperature must be reduced by 7°C.

Program Continuation: Clear alarm with the Delete key or NC START.

2120 NCK fan alarm

Definitions: The fan consists of a 26VDC motor with electronic commutator (rated speed: approx. 8700rpm). The commutator signal is used for speed monitoring, response speed: < 7500rpm.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. The unit with the fan and NCK battery must be replaced.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

2130 5V/24V encoder or 15V D/A converter undervoltage

Definitions: A failure has occurred in the power supply (FM357-2) to the encoder (5V/24V) or D/A converter (+/-15V).

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy: Please inform the authorized personnel/service department. Check the encoder and cable for short-circuits (the fault should not occur when you remove the cable). Check the power feeder line.

Program Continuation: Switch control OFF - ON.

2140 The actual service switch position forces a SRAM to be cleared at the next Power On (general reset active)

Definitions: The initialization switch is currently set to overall reset. This means that the module's SRAM is deleted with the next module reset. The NC data memory is cleared during this operation.

Reactions:
- Alarm display.
- NC not ready.
- Interface signals are set.

Remedy: Reset initialization switch to zero.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

2190 Hardware plug-in module for communication with the digitizer missing

Definitions: MD $MN_ASSIGN_DIGITIZE_TO_CHAN was used to activate the digitizing function by assigning it to a channel. The function requires a hardware module (RS-422 board plugged into the NCU) for communication with the digitizing unit. This module was not found when booting.

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Plug in communications module or cancel channel assignment.

Program Continuation: Switch control OFF - ON.

2192 No NCU link module exists, MD %1 reset

Parameters: %1 = String: MD identifier

Definitions: An attempt was made to activate the NCU link functionality but the hardware is not available. The MD was reset. Only occurs with the NCU link system

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Install the hardware module and activate the function again (MD).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

2193 "Safety Integrated" is not available for link axis %1.

Parameters: %1 = Machine axis index

Definitions: The "Safety Integrated" function is not available for a link axis. Only occurs with the NCU link system

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Use "Safety Integrated" function for local axes only.

Program Continuation: Switch control OFF - ON.

2195 Channel %1 axis %2 high-speed punching/nibbling not possible via link

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: An attempt was made to activate high-speed nibbling or punching for an axis programmed on a different NCU than the drive.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: High-speed nibbling and punching is only supported on one NCU.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

2196 Link axis active and $MN_MM_SERVO_FIFO_SIZE != %1

Parameters: %1 = required value in MD $MN_MM_SERVO_FIFO_SIZE

Definitions: Occurs only with an NCU link system.
- Possible causes of the fault:
- At least one axis is to be distributed via NCU link, then the machine data $MN_MM_SERVO_FIFO_SIZE must be 3 or 4.
- The IPO cycle of this NCU is faster than the link communication cycle, then the machine data $MN_MM_SERVO_FIFO_SIZE must be set to the value proposed in the alarm.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: The machine data $MN_MM_SERVO_FIFO_SIZE must be set to the value proposed in the alarm.

Program Continuation: Switch control OFF - ON.

2900 Reboot is delayed

Definitions: This alarm indicates a delayed reboot.

This alarm only occurs when reboot was carried out by the MMC via PI - "_N_IBN_SS" and MD $MN_REBOOT_DELAY_TIME was set greater than zero.

The alarm can be suppressed with $MN_SUPPRESS_ALARM_MASK BIT 19.

© Siemens AG, 2002. All rights reserved

Reactions: - The NC switches to follow-up mode.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Mode group not ready, also effective for single axes

Remedy: See $MN_REBOOT_DELAY_TIME and $MN_SUPPRESS_ALARM_MASK.

Program Continuation: Switch control OFF - ON.

3000 Emergency stop

Definitions: The EMERGENCY STOP request is applied to the NCK/PLC interface (DB 10, DBX 56.1).

Reactions: - NC Start disable in this channel.
- NC not ready.
- Mode group not ready, also effective for single axes
- Alarm reaction delay is canceled.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Rectify the cause of EMERGENCY STOP and acknowledge EMERGENCY STOP via the PLC/NCK interface (DB 10, DBX 56, Bit 2).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

3001 Internal emergency stop

Definitions: This alarm is not displayed.

Reactions: - NC Start disable in this channel.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.
- Local alarm reaction.

Remedy: No remedy required

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

4000 Channel %1 machine data %2[%3] has gap in axis assignment

Parameters: %1 = Channel number
%2 = String: MD identifier

Definitions: The assignment of a machine axis to a channel by the machine data 20070 AXCONF_MACHAX_USED must be contiguous. At system power-up (Power On) gaps are detected and displayed as an alarm.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. The entries for the indices for the machine axes used in the channels must be contiguous in table $MC_AXCONF_MACHAX_USED. Channel axis gaps must be enabled via $MN_ENABLE_CHAN_AX_GAP.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4001 Channel %1 axis %2 defined for more than one channel via machine data %3

Parameters: %1 = Channel number
%2 = Index: Machine axis number
%3 = String: MD identifier

Definitions: In the channel-specific MD: 20070 AXCONF_MACHAX_USED [CHn, AXm]=x (n ... channel number, m ... channel axis number, x ... machine axis number), several channels were assigned to a machine axis without having a master channel defined for this axis.
There is usually not much point in assigning a machine axis to several channels. In exceptional cases, multiple assignment can be performed if a master channel is defined for this axis. The channel assignment can be performed in accordance with the machining requirements in the NC part program by means of a keyword (yet to be defined in later product versions).

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. In the axis-specific MD 30550 AXCONF_ASSIGN_MASTER_CHAN [AXm]=n (m ... machine axis number, n ... channel number), a master axis was set for the axes that are supposed to be alternately assigned by the NC program to one or the other channel.

Program Continuation: Switch control OFF - ON.

4002 Channel %1 machine data %2[%3] assigns an axis not defined in channel

Parameters: %1 = Channel number
%2 = String: MD identifier
%3 = Index: MD array index

Definitions: Only axes that have been activated in the channel by means of the channel-specific machine data 20070 AXCONF_MACHAX_USED [kx]=m may be declared as geometry axes or transformation axes by means of the MD 20050 AXCONF_GEOAX_ASSIGN_TAB [gx]=k. This also applies to $MC_FGROUP_DEFAULT_AXES (gx: Geometry axis index, kx: Channel axis index, k: Channel axis no., m: Machine axis no.).
Assignment of geometry axes to channel axes
AXCONF_GEOAX_ASSIGN_TAB (includes channel axis no. k):
- Geometry axis index: 0, 1. 0, 2nd channel: 1, 2. 0, 2nd channel: 1
- Geometry axis index: 1, 1. 0, 2nd channel: 2, 2. 0, 2nd channel: 0
- Geometry axis index: 2, 1. 0, 2nd channel: 3, 2. 0, 2nd channel: 3

AXCONF_MACHAX_USED (includes machine axis no. m):
- Channel axis index: 0, 1. 0, 2nd channel: 1, 2. 0, 2nd channel: 4
- Channel axis index: 1, 1. 0, 2nd channel: 2, 2. 0, 2nd channel: 5
- Channel axis index: 2, 1. 0, 2nd channel: 3, 2. 0, 2nd channel: 6
- Channel axis index: 3, 1. 0, 2nd channel: 7, 2. 0, 2nd channel: 0
- Channel axis index: 4, 1. 0, 2nd channel: 8, 2. 0, 2nd channel: 0
- Channel axis index: 5, 1. 0, 2nd channel: 0, 2. 0, 2nd channel: 0
- Channel axis index: 6, 1. 0, 2nd channel: 0, 2. 0, 2nd channel: 0
- Channel axis index: 7, 1. 0, 2nd channel: 0, 2. 0, 2nd channel: 0

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department.
Check and correct either
- $MC_GEOAX_ASSIGN_TAB
- $MC_TRAFO_AXES_IN_X
- $MC_TRAFO_GEOAX_ASSIGN_TAB_X
- $MC_FGROUP_DEFAULT_AXES
- and/or $MC_AXCONF_MACHAX_USED.

Program Continuation: Switch control OFF - ON.

4003 Axis %1 incorrect assignment of master channel in machine data %2

Parameters: %1 = Axis
%2 = String: MD identifier

Definitions: For some applications, it is useful to operate an axis in several channels (C axis or spindle on single spindle or double carriage machines).
The machine axes which are defined in several channels by means of the MD 20070 AXCONF_MACHAX_USED, must be assigned to a master channel with the axis-specific machine data 30550 AXCONF_ASSIGN_MASTER_CHAN.
For axes that are activated in only one channel, the number of this channel or zero must be entered as master channel.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Modify MD 20070: AXCONF_MACHAX_USED and/or MD 30550: AXCONF_ASSIGN_MASTER_CHAN.

Program Continuation: Switch control OFF - ON.

4004 Channel %1 machine data %2 axis %3 defined repeatedly as geometry axis

Parameters: %1 = Channel number
%2 = String: MD identifier
%3 = Axis index

Definitions: An axis may only be defined once as a geometry axis.

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct $MC_GEOAX_ASSIGN_TAB.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4005 Maximum number of axes in channel %1 exceeded. Limit %2

Parameters: %1 = Channel number

%2 = Upper limit for the number of axes in the channel

Definitions: Machine data $MC_AXCONF_MACHAX_USED defines which machine axes can be used in this channel. This simultaneously defines the number of active axes in the channel. This upper limit has been exceeded. Note: The channel axis gaps may cause certain indices of AXCONF_MACHAX_USED to remain unused and therefore do n_o_t count as active channel axes.

Example:

- CHANDATA(2)
- $MC_AXCONF_MACHAX_USED[0] = 7
- $MC_AXCONF_MACHAX_USED[1] = 8
- $MC_AXCONF_MACHAX_USED[2] = 0
- $MC_AXCONF_MACHAX_USED[3] = 3
- $MC_AXCONF_MACHAX_USED[4] = 2
- $MC_AXCONF_MACHAX_USED[5] = 0
- $MC_AXCONF_MACHAX_USED[6] = 1
- $MC_AXCONF_MACHAX_USED[7] = 0

This channel uses the five machine axes 1, 2, 3, 8, 7, i.e. it has 5 active channel axes.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Modify $MC_AXCONF_MACHAX_USED.

Program Continuation: Switch control OFF - ON.

4007 Axis %1 incorrect assignment of master NCU in machine data %2

Parameters: %1 = Axis

%2 = String: MD identifier

Definitions: Machine axes which can be activated on several NCKs through $MN_AXCONF_LOGIC_MACHAX_TAB must be assigned to a master NCU in $MA_AXCONF_ASSIGN_MASTER_NCU. For axes that are activated on only one NCU, the number of this NCU or zero must be entered as master NCU. An assignment can only be made with MA_AXCONF_ASSIGN_MASTER_NCU if the machine axis is also addressed via a channel ($MC_AXCONF_MACHAX_USED+$MN_AXCONF_LOGIC_MACHAX_TAB).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct $MA_AXCONF_ASSIGN_MASTER_NCU and/or $MN_AXCONF_LOGIC_MACHAX_TAB.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4010 Invalid identifier used in machine data %1[%2]

Parameters: %1 = String: MD identifier

%2 = Index: MD array index

Definitions: When determining a name in the NCK tables (arrays) for: machine axes, Euler angles, direction vectors, interpolation parameters and intermediate point coordinates, one of the following syntax rules for the identifier to be entered has been violated:

- The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension (840D: 1-99, FM-NC: 1-9)
- The identifier must begin with 2 arbitrary capital letters but not with $ (reserved for system variable).
- The identifier must not be a keyword of the NC language (e.g. POSA).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Enter the identifier for user-defined names with correct syntax in the displayed MD.

- Machine axes: AXCONF_MACHAX_NAME_TAB
- Euler angles: EULER_ANGLE_NAME_TAB
- Normal vectors: NORMAL_VECTOR_NAME_TAB
- Direction vectors: 10640 DIR_VECTOR_NAME_TAB
- Interpolation parameters: 10650 IPO_PARAM_NAME_TAB
- Intermediate point coordinates: 10660 INTERMEDIATE_POINT_NAME_TAB

Program Continuation: Switch control OFF - ON.

4011 Channel %1 invalid identifier used in machine data %2[%3]

Parameters: %1 = Channel number

%2 = String: MD identifier

%3 = Index: MD array index

Definitions: When defining names in the channel-specific tables for geometry axes and channel axes, one of the following syntax rules for the identifier to be entered has been violated:

- The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension (840D: 1-99, FM-NC: 1-9).
- The identifier must begin with 2 arbitrary capital letters but not with $ (reserved for system variable).
- The identifier must not be a keyword of the NC language (e.g. POSA).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Enter the identifier for user-defined names with correct syntax in the displayed MD

- Geometry axes: 20060 AXCONF_GEOAX_NAME_TAB
- Channel axes: 10000 AXCONF_MACHAX_NAME_TAB

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4012 Invalid identifier used in machine data %1[%2]

Parameters: %1 = String: MD identifier
%2 = Index: MD array index

Definitions: The selected identifier is invalid. Valid identifiers are:
- AX1 - AXn: Machine axis identifiers
- N1AX1 - NnAXm: Link axis identifiers (NCU + machine axis), only occurs with 'NCU link' expansion level!
- C1S1 - CnSm: Container axis identifiers (container + container location). Only occurs with 'axis container' expansion level!

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Use the correct identifier.

Program Continuation: Switch control OFF - ON.

4013 Invalid NCU link configuration by machine data %1 = %2 , on NCU_1 = %3

Parameters: %1 = String: MD identifier
%2 = Index: MD array index
%3 = MD value of master NCU

Definitions: The link module configuration detected on the local NCU is different from the master NCU of the NCU cluster. The link module configuration defines the system clock time, the communication baudrate and the maximum number of message transfer retries.
The following machine data are used for this purpose:
- SYSCLOCK_SAMPL_TIME_RATIO,
- IPO_SYSCLOCK_TIME_RATIO,
- LINK_RETRY_CTR,
- LINK_BAUDRATE_SWITCH,
- SYSCLOCK_CYCLE_TIME

The values of these machine data must be the same on all NCUs.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: The machine data required for the link module configuration must be the same on all NCUs in the cluster.

Program Continuation: Switch control OFF - ON.

4014 Axis %1 defined several times in %2

Parameters: %1 = String: MD identifier
%2 = String: Check and, if necessary, correct the following machine data with reference to the data sheet:

Definitions: An axis was assigned several times.
The axis can be a:
- Machine axis
- Link axis
- Axis in a container location

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Define a correct, unique axis assignment.

Program Continuation: Switch control OFF - ON.

4016 Axis %1 already used by NCU %2

Parameters: %1 = Machine axis index
%2 = NCU number

Definitions: An attempt was made to apply setpoints to one axis from several NCUs. Only occurs with the NCU link system

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Define a correct, unique axis assignment.

Program Continuation: Switch control OFF - ON.

4017 Axis container %1, location %2 already used by NCU %3

Parameters: %1 = Axis container number
%2 = Axis container location
%3 = NCU number

Definitions: A multiple reference to the axis container location has been made via the logical axis table (machine data: MN_AXCONF_LOGIC_MACHAX_TAB). With the NCU link, the multiple reference may also have been made by another NCU in the NCU group.
Example: Container1 location1 was referenced twice incorrectly
- MN_AXCONF_LOGIC_MACHAX_TAB[0] = CT1_SL1
- MN_AXCONF_LOGIC_MACHAX_TAB[6] = CT1_SL1

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct and complete the container location assignments. Check the machine data for the logical axis assignment table (MN_AXCONF_LOGIC_MACHAX_TAB)

Program Continuation: Switch control OFF - ON.

4018 Axis container %1, location %2 not used by any channel

Parameters: %1 = Axis container number
%2 = Axis container location

Definitions: The container location is not referenced by any channel.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Correct and complete the container location assignments. Check machine data MC_AXCONF_MACHAX_USED and MN_AXCONF_LOGIC_MACHAX_TAB.

Program Continuation: Switch control OFF - ON.

4019 Axis container %1 advance not allowed with current status of NCU %2

Parameters: %1 = NCU number

%2 = Axis container number

Definitions: This error only occurs with direct advancing of the container. With direct container advancing, only one channel is allowed to activate the NC language command for advancing the container. In order to ensure this, the other channels must have the reset status and the axes must be stationary.

With NCU link, the above condition applies to all channels of the NCU group.

Error parameters:

- 1 : NC Ready missing
- 16: At least one other channel is active
- 35: AXCT axis is active following axis/spindle
- 36: AXCT axis is active leading axis
- 39: Axis/spindle disable active
- 40: Overlaid motion active for AXCT axis
- 41: Axis replacement active for AXCT axis
- 42: Interpolator active for one axis container axis
- 46: Rotating spindle with different Ipo cycle of NCUs
- 47: New-Config active

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: The program must be canceled with Reset and the zero offset deselected before activating the axis container switch.

Program Continuation: Clear alarm with the RESET key. Restart part program

4020 Identifier %1 used several times in machine data %2

Parameters: %1 = String: Name of identifier

%2 = String: MD identifier

Definitions: When determining a name in the NCK tables (arrays) for: machine axes, Euler angles, direction vectors, interpolation parameters and intermediate point coordinates, an identifier has been used that is already in the control.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Select for the identifier to be entered a character string that is not yet used in the system (max. 32 characters).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

4021 Channel %1 identifier %2 used several times in machine data %3

Parameters: %1 = Channel number
%2 = String: Name of identifier
%3 = String: MD identifier

Definitions: To determine the name in the channel-specific tables for geometry axes and channel axes an identifier already existing in the control has been used.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Select for the identifier to be entered a character string that is not yet used in the system (max. 32 characters).

Program Continuation: Switch control OFF - ON.

4022 Axis container %3 switch not allowed: ext. zero offset active channel %1 axis %2

Parameters: %1 = Channel
%2 = Axis/spindle
%3 = Axis container number

Definitions: The axis container switch enable cannot be given because an external zero offset is active.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: The program must be aborted with the RESET key and the external zero point offset deselected before the container is advanced.

Program Continuation: Clear alarm with the RESET key. Restart part program

4023 Axis container %1 switch not allowed, axis container %2 switch active

Parameters: %1 = Axis container
%2 = Axis container

Definitions: Only one axis container can be rotated at a time.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Program must be canceled with Reset and the program sequences (NCUs, channels) must be synchronized such that only one axis container switch is active at a time.

Program Continuation: Clear alarm with the RESET key. Restart part program

4024 Invalid axis configuration due to missing axis container machine data

Parameters: %1 = NCU number
%2 = Axis container number

Definitions: The axis configuration could not be generated due to missing axis container machine data. This error can only occur as a result of a communication error. The communication failure will be indicated separately by further alarms.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- NC not ready.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct the link communication problems (refer to the other alarm messages).

Program Continuation: Switch control OFF - ON.

4025 Axis container %3 switch not allowed: master/slave active channel %1 axis %2

Parameters: %1 = Channel
%2 = Axis/spindle
%3 = Axis container number

Definitions: It is not possible to enable axis container switch as a master/slave link is active.

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Abort program with the RESET key. If required, disconnect the master - slave coupling.

Program Continuation: Clear alarm with the RESET key. Restart part program

4026 Machine data %1[%2], link axis NC%3_AX%4 not used by any channel

Parameters: %1 = String: MD identifier
%2 = Index: MD array index
%3 = NCU number
%4 = Machine axis number

Definitions: The link axis is not referenced by any channel.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct and complete the logical axis assignments. Check machine data MC_AXCONF_MACHAX_USED and MN_AXCONF_LOGIC_MACHAX_TAB.

Program Continuation: Switch control OFF - ON.

4027 Notice: MD %1 was also changed for the other axes of axis container %2

Parameters: %1 = String: MD identifier
%2 = Axis container number

Definitions: Message to the user indicating that the machine data change for the axis was also performed for all other axes in the same container.

Reactions: - Alarm display.

Remedy: None

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

4028 Notice! The axial MDs of the axes of the axis containers were matched.

Definitions: Note for the user, that the machine data of the axis were matched in the axis containers.
Reactions: - Alarm display.
Remedy: None
Program Continuation: Clear alarm with the RESET key. Restart part program

4029 Notice: the axial MDs in axis container %1 will be matched on the next power-up

Parameters: %1 = Axis container number
Definitions: Message to the user indicating that the machine data of the axes in the axis container will be matched on the next power-up. An axis container allows axes to be exchanged between channels and NCUs. To ensure that no conflicts arise, the axes within the same axis container must have a similar behavior. The first axis in the axis container determines which machine data have to be the same for the other axis in the axis container.
Reactions: - Alarm display.
Remedy: None
Program Continuation: Clear alarm with the Delete key or NC START.

4030 Channel %1 axis identifier missing in machine data %2[%3]

Parameters: %1 = Channel number
%2 = String: MD identifier
%3 = Index: MD array index
Definitions: An axis identifier is expected for the displayed MD in accordance with the axis configuration in the MD 20070 AXCONF_MACHAX_USED and 20050 AXCONF_GEOAX_ASSIGN_TAB.
Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check axis configuration and enter the missing identifier into the MD or, should the axis not exist, specify for this channel axis the machine axis 0 in the channel-specific MD 20070 AXCONF_MACHAX_USED. If this concerns a geometry axis that is not to be used (this applies only for 2-axis machining, e.g. on lathes), then channel axis 0 must be entered additionally in the channel-specific MD 20050 AXCONF_GEOAX_ASSIGN_TAB.
Program Continuation: Switch control OFF - ON.

4031 Channel %1 link axis %2 defined for more than one channel in machine data %3

Parameters: %1 = Channel number
%2 = Index: Axis number for logical axis assignment
%3 = String: MD identifier
Definitions: Occurs only with an NCU link system. The specified axis was defined several times or in several channels in machine data $MC_AXCONF_MACHAX_USED. If an axis is to be defined in several channels, a master channel must be assigned to the axis with the axial machine data $MA_AXCONF_ASSIGN_MASTER_CHAN. This error can only occur with an NCU link axis. The cause of a definition error can also be an NCU link communication failure. The link communication failure must be indicated separately by further alarms.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct the machine data $MC_AXCONF_MACHAX_USED or assign a master channel. In the event of a link communication failure, these error causes have to be remedied first.

Program Continuation: Switch control OFF - ON.

4032 Channel %1 wrong identifier for facing axis in %2

Parameters: %1 = Channel number
%2 = String: MD identifier

Definitions: According to the axis configuration in $MC_GCODE_RESET_VALUES or $MC_DIAMETER_AX_DEF, a facing axis identifier is expected at the specified location.

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Add the correct identifier.

Program Continuation: Switch control OFF - ON.

4033 Notice: NCU link communication still not connected

Definitions: The NCU link communication could not be established due to other active alarms. This is the case, for example, if during boot-up the system detects and modifies incorrect cycle times (see alarm 4110).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Analyze and fix the other alarms and start the control again.

Program Continuation: Switch control OFF - ON.

4034 Local link axis %1 is not allowed for different interpolation cycle time = %2/%3

Parameters: %1 = Axis name
%2 = Local interpolation cycle
%3 = Max. interpolation cycle

Definitions: Local link axes are only permissible on an NCU if the interpolation cycle set corresponds to the slowest interpolation cycle of the interconnected NCU systems.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Remove local link axis (see MN_AXCONF_MACHAX_NAME_TAB and MN_AXCT_AXCONF_ASSIGN_TAB1) or adapt the interpolation cycle (MN_IPO_SYSCLOCK_TIME_RATIO).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4035 Interpolation cycle from NCU%1 = %2 does not match NCU%3 = %4

Parameters: %1 = NCU_number1
%2 = MD value of NCU_number1
%3 = NCU_number2 (with slowest IPO cycle)
%4 = MD value of NCU_number2

Definitions: Occurs only with an NCU link system. The interpolation cycles of the NCUs specified in the alarm do not match one another. The slowest IPO cycle in interconnected NCU systems must be an integral multiple of all configured IPO cycles.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Set a suitable value in MN_IPO_SYSCLOCK_TIME_RATIO for all interconnected NCUs.

Program Continuation: Switch control OFF - ON.

4036 Wrong NCU link configuration by MD %1

Parameters: %1 = String: MD identifier

Definitions: Occurs only with an NCU link system. Different interpolation and position control cycles have been set in the NCUs of the LINK group. This is only allowed if the function FAST-IPO-LINK in MD $MN_MM_NCU_LINK_MASK has been activated.

Notice: For diagnostic purposes, two additional alarm parameters are output together with this alarm.

- 1. 2nd parameter: Position control or IPO cycle time of this NCU
- 2. 2nd parameter: Position control or IPO cycle time of another NCU.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Activate FAST-IPO-LINK function in MN_MM_NCU_LINK_MASK
- Or do not set different position control or IPO cycles on the NCUs (see MN_IPO_SYSCLOCK_TIME_RATIO and MN_POSCTRL_SYSCLOCK_TIME_RATIO).

Program Continuation: Switch control OFF - ON.

4040 Channel %1 axis identifier %2 not consistent with machine data %3

Parameters: %1 = Channel number
%2 = String: Axis identifier
%3 = String: MD identifier
%4 = There are not enough channel axes entered in the MD displayed.

Definitions: The use of the specified axis identifier in the displayed MD is not consistent the channel's axis configuration stated in the MD 20070 AXCONF_MACHAX_USED and MD 20050 AXCONF_GEOAX_ASSIGN_TAB.

Only with active "OEM transformation" compile cycle: There are not enough channel axes entered in the MD displayed.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check and correct the identifier used in the MDs 10000 AXCONF_MACHAX_NAME_TAB, 20080 AXCONF_CHANAX_NAME_TAB and/or 20050 AXCONF_GEOAX_NAME_TAB.

Only with active "OEM transformation" compile cycle: In addition to the specified MD, check and correct MD 24110 TRAFO_AXES_IN_1[n] of the activated OEM transformation according to the function description.

Program Continuation: Switch control OFF - ON.

4045 Channel %1 conflict between machine data %2 and machine data %3

Parameters: %1 = Channel number

%2 = String: MD identifier

%3 = String: MD identifier

Definitions: Using the specified machine data %2 leads to a conflict with machine data %3.

Reactions: - NC not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Mode group not ready, also effective for single axes

Remedy: Correct the specified machine data.

Program Continuation: Switch control OFF - ON.

4050 NC code identifier %1 cannot be reconfigured to %2

Parameters: %1 = String: Old identifier

%2 = String: New identifier

Definitions: Renaming of an NC code was not possible for one of the following reasons:

- The old identifier does not exist
- The new identifier is within another type range.

NC codes/keywords can be reconfigured as long as you stay within the type range.

Type 1: "true" G codes: G02, G17, G33, G64, ...

Type 2: named G codes: ASPLINE, BRISK, TRANS, ...

Type 3: addresses which can be set: X, Y, A1, A2, I, J, K, ALF, MEAS, ...

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Correct machine data 10712: NC_USER_CODE_CONF_NAME_TAB (protection level 1).

The list must be built up as follows:

Even address: Identifier to be modified Following odd address: New identifier

e.g.: NC_USER_CODE_CONF_NAME_TAB [10] = "ROT",
NC_USER_CODE_CONF_NAME_TAB [11] = " " clears the ROT function from the control

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4060 Standard machine data loaded

Definitions: With the next system power-up, the standard MDs are loaded by the system-specific MD 11200 INIT_MD if
- MD buffer voltage has failed or
- after initialization for loading the standard machine data.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. After automatically loading the standard MDs, the individual MDs must be entered or loaded in the relevant system.

Program Continuation: Clear alarm with the RESET key. Restart part program

4062 Backup data loaded

Definitions: The user data saved in the flash memory are loaded to the SRAM.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Load specific machine data again.

Program Continuation: Clear alarm with the RESET key. Restart part program

4065 Buffered memory was restored from backup medium (potential loss of data!)

Definitions: Only occurs with PC-NC. A possible data integrity error was detected in the buffered memory during power-up. The buffered memory was initialized with the last backup copy. Changes in the buffered memory, which have been made since the last backup copy update, have been lost. Backup copies of the buffered memory are updated (on the hard disk) every time the control is shut down normally.

!! Only for 802D: The reason for this procedure is that the backup time is exceeded. Make sure that the required operating time of the control corresponds to the specifications in your Installation & Start-up Guide. The current backup copy of the buffered memory has been created by the last internal data backup via the "Save data" soft key on the HMI.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.

Remedy: Start the control again.

Program Continuation: Switch control OFF - ON.

4066 Buffered memory of FFS restored from backup medium (potential loss of data!)

Definitions: For PC-NC: A possible data integrity error was detected in the FFS memory during power-up. The FFS memory was initialized with the last backup copy. Changes in the FFS memory, which have been made since the last backup copy update, have been lost.

!! Only for PC-NC: Backup copies of the buffered memory are updated (on the hard disk) every time the control is shut down normally.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.

Remedy: Start the control again.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4070 Normalizing machine data has been altered

Definitions: The control uses internal physical units (mm, degrees, s, for paths, velocities, acceleration, etc.). During programming or data storage, some of these values are input and output using different units (rev./min, m/s2, etc.).

The conversion is carried out with the scaling factors that can be entered (system-specific MD array 10230 SCALING_FACTORS USER_DEF[n] (n ... index number 0 - 10), when the corresponding masking bit is set to "1".

If the masking bit is set to "0" then scaling takes place with the internal standard factors.

The following machine data influence the scaling of other MDs:

- 10220: SCALING_USER_DEF_MASK
- 10230: SCALING_FACTORS_USER_DEF
- 10240: SCALING_SYSTEM_IS_METRIC
- 10250: SCALING_VALUE_INCH
- 30300: IS_ROT_AX

If these data are modified, the NCK must be powered up again. Only then will the input of dependent data be performed correctly.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. If the alarm has been displayed after downloading an MD file which is consistent within itself, then the download operation must be repeated with a new NCK power-up. (The file contains scaling-dependent machine data in front of the scaling factors).

Program Continuation: Clear alarm with the Delete key or NC START.

4073 Compile cycle functions define machine data number %1 several times

Parameters: %1 = Machine data number

Definitions: Can only occur when installing compile cycle functions. Two different compile cycle applications use the same machine data number. The machine data which was defined twice is shifted into the free number range above 64000.

Reactions: - Alarm display.

Remedy: The error has no effect on the usability of the machine data and the function of the compile cycle application. To ensure that the compile cycle machine data documentation is correct, you must contact the supplier of the compile cycle. Only the supplier can remedy the error by changing the software.

Program Continuation: Switch control OFF - ON.

4075 Machine data %1 (and maybe others) not altered due to missing permission level %2

Parameters: %1 = String: MD identifier

%2 = Write protection level of the MD

Definitions: On executing a TOA file or when writing data from the part program, an attempt has been made to write an item of data with a higher protection level than the access authorization currently set in the control. The item of data in question has not been written and program execution is continued. This alarm is set only when access violation is detected for the first time.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Set the required access level by means of keyswitch or password entry or delete the machine data concerned from the MD file/part program.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

4076 %1 Machine data could not be altered with permission level %2

Parameters: %1 = Number of MDs
%2 = Preset access authorization

Definitions: On executing a TOA file or when writing data from the part program an attempt has been made to write data with a higher protection level than the access authorization currently set in the control. The data in question have not been written and program execution is continued without hindrance. This alarm is issued on acknowledging the alarm EXBSAL_MD_PERMISSION_DENIED. It can be cleared only with Power On.

Reactions: - Alarm display.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Set the required access level by means of keyswitch or password entry or delete the machine data concerned from the MD file/part program.

Program Continuation: Switch control OFF - ON.

4077 New value %1 of MD %2 not set. Requested %3 bytes too much %4 memory.

Parameters: %1 = New value of machine data
%2 = Machine data number
%3 = Number of bytes requested that exceeded availability
%4 = Type of memory

Definitions: An attempt was made to enter a new value in the specified memory configuration machine data. It was not possible to modify the value, since this would clear the contents of the user memory. This is because the memory requested exceeded the available capacity.

The third parameter specifies the number of bytes by which the maximum user memory was exceeded.

The fourth parameter specifies the type of memory whose limit was exceeded.

- "D" stands for dynamic or non-buffered user memory (this is where the LUD variables are stored and the interpolation buffer size is entered, for example). The capacity of this memory type is defined by the current memory expansion and the value in MD MM_USER_MEM_DYNAMIC (18210).
- "S" stands for static or buffered user memory (this is where part programs, offset data, R parameters, tool data, etc. are stored). This memory type is defined by the current memory expansion and the value in MD MM_USER_MEM_BUFFERED (18230).

Reactions: - Alarm display.

Remedy: If the modification was unintentional, ignore the error message and continue. The alarm has no negative effects. The remedy depends on the access rights and the current memory expansion of the NCK.

- The intended change is not possible -> try again with a smaller value. Observe the change in the number of bytes.
- Buy more memory? This option depends on the model in use.
- The NCK user memory setting may be smaller than possible. The MDs can be changed with appropriate access rights.

Program Continuation: Clear alarm with the Delete key or NC START.

4080 Incorrect configuration of indexing axis in MD %1

Parameters: %1 = String: MD identifier

Definitions: The assignment of a position table to an indexing axis or the contents of a position table contains an error, or the length of a position table has been parameterized with 0.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. 3 MD identifiers are output, depending on the type of error.

1. $MA_INDEX_AX_ASSIGN_TAB (axis-specific MD 30500: The error is due to multiple assignment of a position table (NCK MD 10910/10930 INDEX_AX_POS_TAB_n) to axes with different types (linear/rotary axis).

2. $MN_INDEX_AX_POS_TAB_n (NCK MD 10910/10930): The contents of the displayed table n contain an error.

- The entered positions must be arranged in increasing size.
- A particular position must not be set more than once.
- If the table is assigned to one or several modulo axes, then the contents must be within the 0 to < 360 degree range.

3. $MN_INDEX_AX_LENGTH_POS_TAB_n (NCK MD 10900/10920): The length of the displayed position table n was specified with 0.

Program Continuation: Clear alarm with the RESET key. Restart part program

4090 Too many errors during power-up

Definitions: More than <n> errors occurred during control power-up.

Reactions: - Alarm display.
- NC Start disable in this channel.

Remedy: Set the machine data correctly.

Program Continuation: Switch control OFF - ON.

4100 System cycle time/scan time divider corrected for digital drive

Definitions: The machine data 10050 SYSCLOCK_CYCLE_TIME (system clock cycle time) and/or MD 10080 SYSCLOCK_SAMPL_TIME_RATIO (dividing factor of the position control cycle for actual value acquisition) have been corrected. If this requirement cannot be satisfied with the entered values (e.g. because the system clock cycle time is not a multiple of 31.25 ms), then the system clock cycle time is automatically expanded until the drive clock cycle time lies within the 31.25 ms grid.

The modifications were so made that, due to the selection of the system clock cycle time in MD 10050 SYSCLOCK_CYCLE_TIME, the programmable hardware divider 1 was readjusted in such a way that the selected time and the basic drive cycle result in a 31.25 ms grid. If this requirement is unfeasible (e.g. because the system clock cycle is not a multiple of 31.25 ms), the system clock cycle is automatically increased until the basic drive cycle is in a 31.25 ms grid.

The new value of the SYSCLOCK_CYCLE_TIME can be obtained from the MD 10050.

The position control cycle can be set with the following gradations:

- up to 4 ms: 125 µs step
- up to 8 ms: 250 µs step
- up to 16 ms: 0.5 ms step
- up to 32 ms: 1 ms step

Reactions: - Alarm display.

Remedy: No remedial measures are required. The alarm display can be canceled with Reset.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

4101 Position control cycle for digital drive reduced to %1 ms

Parameters: %1 = String (time in ms)

Definitions: The position control clock divisor in the NCK MD 10060 POSCTRL_SYSCLOCK_TIME_RATIO was set such that a position control cycle time of more than 16 ms resulted. The boundary value for the drive actuator 611D is however 16 ms.

Reactions: - Alarm display.

Remedy: No remedial measures are required. The alarm display is canceled with Reset.

Program Continuation: Switch control OFF - ON.

4102 Default values for drive cycle times differ

Definitions: External control modules of the 611D bus and the controls within the CCU3 module have different default values for the current and speed control cycle times.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: External control modules of the 611D bus and the controls within the CCU3 module have different default values for the current and speed control cycle times.

Check the specified values and modify accordingly (see MD_CURRCTRL_CYCLE_TIME and MD_SPEEDCTRL_CYCLE_TIME).

Program Continuation: Switch control OFF - ON.

4110 IPO factor increased to %1

Parameters: %1 = String (new IPO cycle time)

Definitions: The IPO cycle divisor was set to a value which was not an integral multiple of the position control cycle divisor. The divisor (MD 10070 IPO_SYSCLOCK_TIME_RATIO) was increased. IPO_SYSCLOCK_TIME_RATIO has been modified on systems with PROFIBUS DP because of the modified DP cycle in SDB1000 (MD 10050 SYSCLOCK_CYCLE_TIME).

Reactions: - Alarm display.

Remedy: Machine data 10070 IPO_SYSCLOCK_TIME_RATIO has been modified.

Program Continuation: Clear alarm with the RESET key. Restart part program

4111 PLC cycle increased to %1 ms

Parameters: %1 = String (new PLC cycle time)

Definitions: The PLC cycle divisor was set to a value which was not an integral multiple of the IPO cycle divisor. The divisor (MD 10074 PLC_IPO_TIME_RATIO) has been increased. MD 10074 PLC_IPO_TIME_RATIO has been modified on systems with PROFIBUS DP because of the modified DP cycle in SDB1000 (MD 10050 SYSCLOCK_CYCLE_TIME).

Reactions: - Alarm display.

Remedy: Machine data 10074 PLC_IPO_TIME_RATIO has been modified.

Program Continuation: Clear alarm with the RESET key. Restart part program

4112 Servo cycle changed to %1 ms

Parameters: %1 = String (new servo cycle time)

Definitions: MD 10060 POSCTRL_SYSCLOCK_TIME_RATIO has been modified on systems with PROFIBUS DP because of the modified DP cycle in SDB1000 (10050 SYSCLOCK_CYCLE_TIME).

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Machine data 10060 POSCTRL_SYSCLOCK_TIME_RATIO RATIO has been modified.
Program Continuation: Clear alarm with the RESET key. Restart part program

4113 Sysclock cycle changed to %1 ms

Parameters: %1 = String (new PLC cycle time)
Definitions: MD 10050 SYSCLOCK_CYCLE_TIME has been modified on systems with PROFIBUS DP because of the modified DP cycle in SDB1000.
Reactions: - Alarm display.
Remedy: Machine data 10050 SYSCLOCK_CYCLE_TIME has been modified.
Program Continuation: Clear alarm with the RESET key. Restart part program

4114 Error in DP cycle of SDB1000

Parameters: %1 = String (new PLC cycle time)
Definitions: The DP cycle in SDB1000 contains an error and cannot be set. The default value of $MN_SYSCLOCK_CYCLE_TIME is set.
Reactions: - Alarm display.
Remedy: Correct SDB1000
Program Continuation: Switch control OFF - ON.

4115 Time ratio communication to Ipo changed to %1

Parameters: %1 = String (new PLC cycle time)
Definitions: The value of the machine data 10072 has been adapted. This can only occur, if the value of the machine data is smaller than one and the time thus calculated is no multiple of the position control cycle.
Reactions: - Alarm display.
Remedy: The machine data $MN_COM_IPO_TIME_RATIO has been adapted. Please check to ensure that the calculated value is correct.
Program Continuation: Clear alarm with the RESET key. Restart part program

4150 Channel %1 invalid M function subprogram call configured

Parameters: %1 = Channel number
Definitions: The machine data $MN_M_NO_FCT_CYCLE[n] or $MN_M_NO_FCT_CYCLE_PAR contains invalid configuration data: An M function, which is used by the system and can not be replaced by a subprogram call has been specified in the machine data $MN_M_NO_FCT_CYCLE[n] for the configuration of the subprogram call via M function:

- M0 to M5,
- M17, M30,
- M19, M40 to M45,
- M function for selecting spindle/axis mode according to $MC_SPIND_RIGID_TAPPING_M_NR (default: M70),
- M functions for nibbling/punching as configured in $MC_NIBBLE_PUNCH_CODE if activated by $MC_PUNCHNIB_ACTIVATION.
- Also M96 to M99 for applied external language ($MN_MM_EXTERN_LANGUAGE).

The machine data $MN_M_NO_FCT_CYCLE_PAR contains an invalid array index of $MN_M_NO_FCT_CYCLE[n]. Currently, the values 0 to 9 are permissible. The affected machine data is reset to the default value -1. This deactivates the function.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Configure an M function in the machine data $MN_M_NO_FCT_CYCLE[n] that is not assigned by the system, or configure a permissible array index in the machine data $MN_M_NO_FCT_CYCLE_PAR.

Program Continuation: Switch control OFF - ON.

4152 Illegal configuration of the "Block display with absolute values" function

Definitions: The "Block display with absolute values" function has been illegally parameterized:

- An illegal block length has been set with $MC_MM_ABSBLOCK:

While ramping up, the machine data will be checked for the following value range: 0, 1, 128 to 512

- An invalid display range has been set with $MC_MM_ABSBLOCK_BUFFER_CONF[].

While ramping up, the machine data will be checked for the following upper and lower limits:

- 0 <= $MC_MM_ABSBLOCK_BUFFER_CONF[0] <= 8
- 0 <= $MC_MM_ABSBLOCK_BUFFER_CONF[1] <= ($MC_MM_IPO_BUFFER_SIZE + $MC_MM_NUM_BLOCKS_IN_PREP). Alarm 4152 is issued if the limits are violated.

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Configure block length/display range within the permissible limits.

Program Continuation: Switch control OFF - ON.

4160 Channel %1 invalid M function number configured for spindle switchover

Parameters: %1 = Channel number

Definitions: An M function was specified in machine data $MC_SPIND_RIGID_TAPPING_M_NR in order to configure the M function number for spindle switchover. The M function number is assigned by the system and cannot be used for the switchover (M1 to M5, M17, M30, M40 to M45).

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Configure an M function which is not used by the system (M1 to M5, M17, M30, M40 to M45) in machine data $MC_SPIND_RIGID_TAPPING_M_NR.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4170 Invalid M function number for channel synchronisation assigned

Definitions: An M number between 0 and 99 has been specified In machine data $MN_EXTERN_CHAN_SYNC_M_NR_MIN or $MN_EXTERN_CHAN_SYNC_M_NR_MAX for the configuration of the M number range for channel synchronization in ISO2/3 mode or the machine data $MN_EXTERN_CHAN_SYNC_M_NR_MAX is smaller than $MN_EXTERN_CHAN_SYNC_M_NR_MIN.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data $MN_EXTERN_CHAN_SYNC_M_NR_MIN and $MN_EXTERN_CHAN_SYNC_M_NR_MAX.

Program Continuation: Switch control OFF - ON.

4180 Invalid M function number assigned to enable ASUP

Definitions: An invalid M function number has been assigned for activation of ASUP. An illegal M number has been assigned in machine data $MN_EXTERN_M_NO_SET_INT or $MN_EXTERN_M_NO_DISABLE_INT for the configuration of the M number range for activation/deactivation of the interrupt program.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data $MN_EXTERN_ M_NO_SET_INT and $MN_EXTERN_M_NO_DISABLE_INT.

Program Continuation: Switch control OFF - ON.

4181 Channel %1 invalid assignment of an M auxiliary function number

Parameters: %1 = Channel number

Definitions: In machine data $MC_AUXFU_ASSOC_M0_VALUE or $MC_AUXFU_ASSOC_M1_VALUE, a number has been specified for the configuration of a new predefined M function which is occupied by the system, and cannot be used for an assignment. (M0 to M5, M17, M30, M40 to M45).

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Configure an M function in machine data $MC_AUXFU_ASSOC_M0_VALUE or $MC_AUXFU_ASSOC_M1_VALUE which is not occupied by the system (M1 to M5, M17, M30, M40 to M45).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4182 Channel %1 invalid M auxiliary function number in %2%3, MD reset

Parameters: %1 = Channel number
%2 = MD identifier
%3 = If required, MD index

Definitions: In the specified machine data, a number has been specified for the configuration of an M function which is occupied by the system, and cannot be used for an assignment. (M0 to M5, M17, M30, M40 to M45 and also M98, M99 with applied ISO dialect). The value set by the user has been reset to the default value by the system.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Configure an M function in the specified machine data which is not occupied by the system (M0 to M5, M17, M30, M40 to M45 and also M98, M99 with applied ISO dialect).

Program Continuation: Clear alarm with the RESET key. Restart part program

4183 Channel %1 M auxiliary function number %2 used several times (%3 and %4)

Parameters: %1 = Channel number
%2 = M auxiliary function number
%3 = MD identifier
%4 = MD identifier

Definitions: In the specified machine data, a number has been used several times for the configuration of an M function.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check the specified machine data and create a unique assignment of M auxiliary function numbers.

Program Continuation: Switch control OFF - ON.

4184 Channel %1 illegally predefined auxiliary function in %2%3, MD reset

Parameters: %1 = Channel number
%2 = MD identifier
%3 = If required, MD index

Definitions: In the specified machine data, a predefined auxiliary function has been illegally configured.
The value set by the user has been reset to the default value by the system.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Configure a valid value in the specified machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

4185 Channel %1 illegal auxiliary function configured %2 %3 %4

Parameters: %1 = Channel number
%2 = Type of auxiliary function
%3 = Extension
%4 = Auxiliary function value

Definitions: An auxiliary function has been illegally configured.
Predefined auxiliary functions cannot be reconfigured by user-defined auxiliary functions.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Reconfigure the auxiliary function.

Program Continuation: Clear alarm with the RESET key. Restart part program

4200 Channel %1 geometry axis %2 must not be declared a rotary axis

Parameters: %1 = Channel number
%2 = Axis name

Definitions: The geometry axes represent a Cartesian coordinate system and therefore the declaration of a geometry axis as rotary axis leads to a definition conflict.

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Remove rotary axis declaration for this machine axis.
For this purpose, the geometry axis index for the displayed geometry axis must be determined by means of the channel-specific machine data array 20060 AXCONF_GEOAX_NAME_TAB. The channel axis number is stored with the same index in the channel-specific MD array 20050 AXCONF_GEOAX_ASSIGN_TAB. The channel axis number minus 1 provides the channel axis index under which the machine axis number is found in the channel-specific MD array 20070 AXCONF_MACHAX_USED.

Program Continuation: Switch control OFF - ON.

4210 Channel %1 spindle %2 declaration as rotary axis missing

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: If a machine axis is to be operated as a spindle, this machine axis must be declared as a rotary axis.

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set rotary axis declaration for this machine axis in the axis-specific MD 30300 IS_ROT_AX.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4215 Channel %1 spindle %2 declaration as modulo axis missing

Parameters: %1 = Channel number
%2 = Axis name, spindle number
Definitions: The spindle functionality requires a modulo axis (positions in [deg],.).
Reactions: - Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
Remedy: Please inform the authorized personnel/service department. Set MD "ROT_IS_MODULO".
Program Continuation: Switch control OFF - ON.

4220 Channel %1 spindle %2 declared repeatedly

Parameters: %1 = Channel number
%2 = Axis name, spindle number
Definitions: The spindle number exists more than once in the channel.
Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
Remedy: Please inform the authorized personnel/service department. The spindle number is stored in the axis-specific MD array 35000 SPIND_ASSIGN_TO_MACHAX. The channel to which this machine axis/spindle is assigned is listed in the machine axis index. (The machine axis number is given in the channel-specific MD array 20070 AXCONF_MACHAX_USED).
Program Continuation: Switch control OFF - ON.

4225 Channel %1 axis %2 declaration as rotary axis missing

Parameters: %1 = Channel number
%2 = Axis name, axis number
Definitions: The modulo functionality requires a rotary axis (positions in [deg],.).
Reactions: - Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
Remedy: Please inform the authorized personnel/service department. Set MD "IS_ROT_AX".
Program Continuation: Switch control OFF - ON.

4230 Channel %1 data alteration from external not possible in current channel state

Parameters: %1 = Channel number
Definitions: It is not allowed to enter this data while the part program is being executed (e.g. setting data for working area limitation or for dry run feedrate).
Reactions: - Alarm display.
Remedy: The data to be entered must be altered before starting the part program.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

4240 Runtime overflow for IPO cycle or position controller cycle, IP %1

Parameters: %1 = Program location

Definitions: The settings for the interpolation and position control cycle were modified before the last power-up such that too little computing time is now available for the requisite cyclic task.

The alarm occurs immediately after power-up if too little runtime is available even when the axes are stationary and the NC program has not started. However, task overflow can occur only when computation-intensive NC functions are called during program execution.

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- Alarm reaction delay is canceled.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Take greater care when optimizing the clock times NCK MD 10050 SYSCLOCK_CYCLE_TIME, MD 10060 POSCTRL_SYSCLOCK_TIME_RATIO and/or MD 10070 IPO_SYSCLOCK_TIME_RATIO.

The test should be performed with an NC program that represents the worst case. For safety, a margin of 15 to 25% should be added to the times determined in this way.

Program Continuation: Switch control OFF - ON.

4250 FastPlcCom functionality not available

Definitions: This alarm indicates that the PLC provides the None FastPlcCom functionality during start-up although this functionality is requested by the NCK.

Reactions: - Alarm display.

Remedy: Retrofit the PLC with the FastPlcCom functionality or deactivate the FastPlcCom functionality by means of NCK machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

4252 PLCIO read error: %1

Parameters: %1 = PLCIO error code

Definitions: This alarm indicates that errors occured when reading the PLCIO with the FastPlcCom functionality.

Reactions: - Alarm display.

Remedy: Correct the machine data or check the PLC hardware configuration.

Program Continuation: Clear alarm with the RESET key. Restart part program

4254 PLCIO write error: %1

Parameters: %1 = PLCIO error code

Definitions: This alarm indicates that errors occured when writing on the PLCIO with the FastPlcCom functionality.

Reactions: - Alarm display.

Remedy: Correct the machine data or check the PLC hardware configuration.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

4260 Machine data %1 illegal

Parameters: %1 = String: MD identifier

Definitions: Selected cam pair not activated by MD $MN_SW_ASSIGN_TAB or several cam pairs selected.

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Activate the cam pair or select only one cam pair.

Program Continuation: Switch control OFF - ON.

4270 Machine data %1 assigns not activated NCK input/output byte %2

Parameters: %1 = String: MD identifier

%2 = Index

Definitions: The specified machine data assigns a digital input/output byte or an analog input/output signal the processing of which has not been activated to an NC function.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Correct machine data. Activate required inputs/outputs via MDs:
- $MN_FASTIO_DIG_NUM_INPUTS
- $MN_FASTIO_DIG_NUM_OUTPUTS
- $MN_FASTIO_ANA_NUM_INPUTS
- $MN_FASTIO_ANA_NUM_OUTPUTS

Activation of fast inputs/outputs does not require the corresponding hardware configuration to be available at the control. All functions using fast inputs/outputs can also be made use of by the PLC specification/modification defined in the VDI interface, if the response time requirements are reduced accordingly.

Activated inputs/outputs increase the computation time requirement of the interpolation cycle because the PLC manipulation signals are handled cyclically. Note: Deactivate any inputs/outputs not in use.

Program Continuation: Switch control OFF - ON.

4275 Machine data %1 and %2 both assign the same NCK output byte no. %3 several times

Parameters: %1 = String: MD identifier

%2 = String: MD identifier

%3 = No. of output

Definitions: The specified machine data assign two NC functions to the same digital/analog output.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Correct machine data.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

4280 Assignment of NCK input/output byte via MD %1[%2] does not match hardware configuration

Parameters: %1 = String: MD identifier
%2 = MD array index

Definitions: When booting, the required input/output module was not found at the slot specified in the MD.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check hardware and correct the MD if necessary. Note: Monitoring of the hardware configuration is performed independently of the number of activated inputs/outputs (MD 10300 - 10360 FASTIO_ANA(DIG)_NUM_INPUTS(OUTPUTS))

Program Continuation: Switch control OFF - ON.

4282 Hardware of external NCK outputs assigned repeatedly

Definitions: Several outputs have been configured on the same hardware byte.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Alter MD 10364 HW_ASSIGN_DIG_FASTOUT or MD 10364 HW_ASSIGN_ANA_FASTOUT.

Program Continuation: Switch control OFF - ON.

4285 Error on terminal block %1, error code %2

Parameters: %1 = Number of terminal block (1 ... 4)
%2 = Error code

Definitions: An error occurred on terminal block no. %1 (sign-of-life failure, I/O module removed in current operation, etc.). All possible errors which can lead to this alarm are not yet known (and will be completed later). Further information together with a description of the error code and its meaning will be provided at a later date.
Error code 1: Sign-of-life failure from terminal block
Error code 10: Sign-of-life failure NC

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check hardware.

Program Continuation: Switch control OFF - ON.

4290 Local P-bus sign-of-life monitoring

Definitions: The COM computer must alter the sign-of-life on the local P-bus in each SERVO cycle. Monitoring for alteration takes place in the IPO cycle. If the sign of life has not altered, this alarm is triggered.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check hardware.

Program Continuation: Switch control OFF - ON.

4291 Module in local P-bus slot %1 error codes %2 %3 %4 %2 %3 %4

Parameters: %1 = Slot number
%2 = Error code
%3 = Error code
%4 = Error code

Definitions: The module on the specified slot has signaled a diagnostics alarm. The error code reported corresponds to the AS300 documentation.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Check hardware.

Program Continuation: Switch control OFF - ON.

4300 Declaration in MD %1 is not allowed for geometry axis/spindle %2.

Parameters: %1 = String: MD identifier
%2 = Axis name, spindle number

Definitions: Geometry axes and spindles cannot be operated as concurrent positioning axes.

Reactions: - NC not ready.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reset MD 30450 IS_CONCURRENT_POS_AX for the axis concerned.

Program Continuation: Switch control OFF - ON.

4310 Declaration in MD %1 index %2 is not allowed.

Parameters: %1 = String: MD identifier
%2 = Index in MD array

Definitions: The machine data values must be written in the array in ascending order.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Correct the MD.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

4320 Axis %1 function %2 %3 and %4 not allowed

Parameters: %1 = String: Axis identifier
%2 = String: MD identifier
%3 = String: Bit
%4 = String: MD identifier

Definitions: The functions declared by the specified machine data cannot simultaneously be active for one axis.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Deactivate one of the functions.

Program Continuation: Switch control OFF - ON.

4334 Channel %1 The amount of fine correction in parameter %2 of the orientable toolholder %3 is too large

Parameters: %1 = Channel number
%2 = Invalid parameter of the orientable toolholder
%3 = Number of the orientable toolholder

Definitions: The maximum permissible value of the fine correction in an orientable toolholder is limited by the machine data $MC_TOCARR_FINE_LIM_LIN for linear variables, and by the machine data $MC_TOCARR_FINE_LIM_ROT for rotary variables. The alarm can only occur if the setting data $SC_TOCARR_FINE_CORRECTION is not equal to zero.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter a valid fine correction value.

Program Continuation: Clear alarm with the RESET key. Restart part program

4336 Channel %1 orientable toolholder no. %2 for orientation transformation %3 does not exist

Parameters: %1 = Channel number
%2 = Number of the orientable toolholder
%3 = Number of the orientation transformation that is to be parameterized with the orientable toolholder

Definitions: The orientable toolholder, with whose data the orientation transformation is to be parameterized (see machine data $MC_TRAFO5_TCARR_NO_1/2), does not exist.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter a valid tool-carrier number.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

4338

Channel %1 invalid transformation type '%2' in toolholder %3 for orientation transformer %4

Parameters: %1 = Channel number
%2 = Transformer type
%3 = Number of the orientable toolholder
%4 = Number of the orientation transformation that is to be parameterized with the orientable toolholder

Definitions: The parameters of the orientation transformation are taken over from the data of an orientable toolholder. This orientable toolholder contains an invalid transformation type. (Types T, P and M are permissible).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter a valid transformation type.

Program Continuation: Clear alarm with the RESET key. Restart part program

4340

Channel %1 block %2 invalid transformation type in transformation no. %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Transformation number

Definitions: An invalid, i.e. undefined, number was entered in one of the machine data TRAFO_TYPE_1 ... TRAFO_TYPE_8. This alarm also occurs if a certain type of transformation is only impossible on the type of control used (e.g. 5-axis transformation on a SINUMERIK 802D).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter a valid transformation type.

Program Continuation: Clear alarm with the RESET key. Restart part program

4341

Channel %1 block %2 no data set available for transformation no. %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Transformation number

Definitions: Only a limited number of machine data sets (usually 2) is available for each related group of transformations (e.g. orientation transformations, Transmit, Tracyl, etc.). This alarm is output if an attempt is made to set more transformations from a group.
Example:
Two orientation transformations are allowed. The machine data contains e.g.:
TRAFO_TYPE_1 = 16 ; 1st orientation transformation
TRAFO_TYPE_2 = 33 ; 2nd orientation transformation
TRAFO_TYPE_3 = 256 ; 1st transmit transformation
TRAFO_TYPE_4 = 20 ; 3rd orientation transformation ==> This entry triggers alarm

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter valid machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

4342 Channel %1 invalid machine data for general 5-axis transformation error no. %2

Parameters: %1 = Channel number
%2 = Error type

Definitions: The machine data which describe the axis directions and the base orientation for the general 5-axis transformation are invalid. The error parameter displayed specifies the cause of the alarm:
- 1: The first axis (TRAFO5_AXIS1_*) is not defined (all three entries of the vector are 0)
- 2: The second axis (TRAFO5_AXIS2_*) is not defined (all three entries of the vector are 0)
- 3: The base orientation (RAFO5_BASE_ORIEN_*) is not defined (all three entries of the vector are 0)
- 4: The first and second axis are (virtually) parallel

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Set valid machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

4343 Channel %1 attempt made to change the machine data of an active transformation.

Parameters: %1 = Channel number

Definitions: An attempt was made to change the machine data of an active transformation and to activate the machine data with RESET or NEWCONFIG.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Stop on alarm at block end.

Remedy: Set valid machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

4345 Channel %1 invalid configuration in chained transformation no. %2

Parameters: %1 = Channel number
%2 = Transformation number

Definitions: A chained transformation is incorrectly configured (machine data $MC_TRACON_CHAIN_1 or $MC_TRACON_CHAIN_2). The following causes for the error are possible:
- The list of transformations to be chained starts with a 0 (at least one entry not equal to zero is required).
- The list of transformations to be chained contains the number of a transformation which does not exist.
- The number of a transformation in the list is greater than or equal to the number of the chained transformation. Example: The cascaded transformation is the fourth transformation in the system, i.e. $MC_TRAFO_TYPE_4 = 8192. In this case, only values 1, 2 or 3 may be entered in the associated list (e.g. $MC_TRACON_CHAIN_1[...]).
- The chaining setting is invalid. The following restrictions currently apply. A maximum of two transformations can be chained. The first transformation must be an orientation transformation, transmit, peripheral curve transformation or inclined axis. The second transformation must be the inclined axis transformation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Set a valid transformation chain.
Program Continuation: Clear alarm with the RESET key. Restart part program

4346 Channel %1 invalid geoaxis assignment in machine data %2[%3]

Parameters: %1 = Channel number
%2 = Name of machine data
%3 = Transformation number
Definitions: Machine data TRAFO_GEOAX_ASSIGN_TAB_X contains an invalid entry. The following causes for the error are possible:
- The entry references a channel axis which does not exist.
- The entry is zero (no axis) but the transformation needs the relevant axis as a geometry axis.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Correct the entry in TRAFO_GEOAX_ASSIGN_TAB_X or TRAFO_AXES_IN_X.
Program Continuation: Clear alarm with the RESET key. Restart part program

4347 Channel %1 invalid channel axis assignment in machine data %2[%3]

Parameters: %1 = Channel number
%2 = Name of machine data
%3 = Transformation number
Definitions: Machine data TRAFO_AXIS_IN_X contains an invalid entry. The following causes for the error are possible:
- The entry references a channel axis which does not exist.
- The entry is zero (no axis) but the transformation needs the relevant axis as a channel axis.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Correct the entry in TRAFO_AXES_IN_X.
Program Continuation: Clear alarm with the RESET key. Restart part program

4350 Channel %1 axis identifier %2 machine data %3 not consistent with machine data %4

Parameters: %1 = Channel number
%2 = String: Axis identifier
%3 = String: MD identifier
%4 = String: MD identifier
Definitions: MD 32410 JOG_AND_POS_JERK_ENABLE (jerk limitation) and MD 35240 ACCEL_TYPE_DRIVE (acceleration reduction) have been defined as the initial setting for an axis. However, the two functions cannot be activated at the same time for one axis.
Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Resetting of 32410 JOG_AND_POS_JERK_ENABLE or 35240 ACCEL_TYPE_DRIVE.

Program Continuation: Switch control OFF - ON.

4400 MD alteration will cause reorganisation of buffered memory (loss of data!)

Definitions: A machine data has been altered that configures the buffered memory. If the NCK powers up with the altered data, this will lead to reorganization of the buffered memory and thus to the loss of all buffered user data (part programs, tool data, GUD, leadscrew error compensation, ...)

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. If the control includes user data that has not yet been saved, then a data backup must be performed before the next NCK power-up. By manually resetting the altered MD to the value it had before the last power-up, reorganization of the memory can be avoided.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

4502 Channel %1 anachronism %2(%3) -> %4

Parameters: %1 = Channel number

%2 = String: MD identifier

%3 = String: MD identifier

%4 = String: MD identifier

Definitions: Previously, in $MC_RESET_MODE_MASK Bit4 and Bit5, the reset behavior of the 6th or 8th G group was determined. This setting is now made in $MC_GCODE_RESET_MODE.

In order to ensure compatible handling of "old" data backups, the "old" values are taken from $MC_RESET_MODE_MASK and entered in $MC_GCODE_RESET_MODE.

Reactions: - Alarm display.

Remedy: -

Program Continuation: Clear alarm with the Delete key or NC START.

5000 Communication job not executable %1

Parameters: %1 = Reference to which resources are no longer available.

Definitions: The communication job (data exchange between NCK and MMC, e.g.: loading an NC part program) cannot be executed because there is insufficient memory space. Cause: Too many communication jobs in parallel.

Reactions: - Alarm display.

Remedy:
- Reduce the number of communication jobs taking place at the same time or increase $MN_MM_NUM_MMC_UNITS
- Restart communication job.

Please inform the authorized personnel/service department. No remedial measures are possible - the operation triggering the alarm message has to be repeated. Clear the alarm display with Cancel.

Program Continuation: Clear alarm with the Delete key or NC START.

6000 Memory reorganized using standard machine data

Definitions: The memory management was not able to allocate the NC user memory with the values in the machine data. Because the total memory available is provided as dynamic and static memory for the NC user (e.g. for macro definitions, user variables, number of tool offsets, number of directories and files etc.) and therefore its size is not adequate.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Redefine the NC memory structure!

A specific machine data for NC user memory allocation cannot be given as the cause of the alarm. Therefore, the MD initiating the alarm must be determined on the basis of the default values in the machine data by changing the user-specific memory structure step by step.

Usually, not just one machine data has been chosen too large and therefore it is advisable to reduce the memory area by a certain proportion in several MDs.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

6010 Channel %1 data block %2 not or not completely created, error code %3

Parameters: %1 = Channel number

%2 = String (block name)

%3 = Internal error code

Definitions: Data management has detected an error in power-up. The specified data block may not have been created. The error number specifies the type of error. If the error number >100000, then there is a fatal system error. Otherwise, the user memory area was made too small. In this case the (user) error codes have the following meaning:

- Error number 1: No memory space available
- Error number 2: Maximum possible number of symbols exceeded
- Error number 3: Index 1 outside of valid value range
- Error number 4: Name in channel already exists
- Error number 5: Name in NCK already exists

If the alarm occurs after cycle programs, macro definitions or definitions for global user data (GUD) have been introduced, the machine data for the NC user memory configuration have been incorrectly configured. In all other cases, changes to machine data that are already correct lead to errors in the user memory configuration.

The following block names (2nd parameter) are known in the NCK (all system and user data blocks; in general, only problems in the user data blocks can be remedied by user intervention:

- _N_NC_OPT - System internal: option data, NCK global
- _N_NC_SEA - System internal: setting data, NCK global
- _N_NC_TEA - System internal: machine data, NCK global
- _N_NC_CEC - System internal: 'cross error compensation'
- _N_NC_PRO - System internal: protection zones, NCK global
- _N_NC_GD1 - User: 1. GUD block defined by _N_SGUD_DEF, NCK global
- _N_NC_GD2 - User: 2. GUD block defined by _N_MGUD_DEF, NCK global
- _N_NC_GD3 - User: 3. GUD block defined by _N_UGUD_DEF, NCK global
- _N_NC_GD4 - User: 4. GUD block defined by _N_GUD4_DEF, NCK global
- _N_NC_GD5 - User: 5. GUD block defined by _N_GUD5_DEF, NCK global
- _N_NC_GD6 - User: 6. GUD block defined by _N_GUD6_DEF, NCK global
- _N_NC_GD7 - User: 7. 7th GUD block defined by _N_GUD7_DEF, NCK global
- _N_NC_GD8 - User: 8. GUD block defined by _N_GUD8_DEF, NCK global
- _N_NC_GD9 - User: 9. GUD block defined by _N_GUD9_DEF, NCK global
- _N_NC_MAC - User: Macro definitions
- _N_NC_FUN - User: Cycle programs

© Siemens AG, 2002. All rights reserved

- _N_CHc_OPT - System internal: option data, channel-specific
- _N_CHc_SEA - System internal: setting data, channel-specific
- _N_CHc_TEA - System internal: machine data, channel-specific
- _N_CHc_PRO - System internal: protection zones, channel-specific
- _N_CHc_UFR - System internal: frames, channel-specific
- _N_CHc_RPA - System internal: arithmetic parameter, channel-specific
- _N_CHc_GD1 - User: 1. GUD block defined by _N_SGUD_DEF, channel-specific
- _N_CHc_GD2 - User: 2. GUD block defined by _N_MGUD_DEF, channel-specific
- _N_CHc_GD3 - User: 3. GUD block defined by _N_UGUD_DEF, channel-specific
- _N_CHc_GD4 - User: 4. GUD block defined by _N_GUD4_DEF, channel-specific
- _N_CHc_GD5 - User: 5. GUD block defined by _N_GUD5_DEF, channel-specific
- _N_CHc_GD6 - User: 6. GUD block defined by _N_GUD6_DEF, channel-specific
- _N_CHc_GD7 - User: 7. 7th GUD block defined by _N_GUD7_DEF, channel-specific
- _N_CHc_GD8 - User: 8. GUD block defined by _N_GUD8_DEF, channel-specific
- _N_CHc_GD9 - User: 9. GUD block defined by _N_GUD9_DEF, channel-specific
- _N_AXa_OPT - System internal: option data, axial
- _N_AXa_SEA - System internal: setting data, axial
- _N_AXa_TEA - System internal: machine data, axial
- _N_AXa_EEC - System internal: leadscrew error compensation data, axial
- _N_AXa_QEC - System internal: quadrant error compensation data, axial
- _N_TOt_TOC - System internal: toolholder data, TOA-specific
- _N_TOt_TOA - System internal: tool data, TOA-specific
- _N_TOt_TMA - System internal: magazine data, TOA-specific

c = Channel number

a = Machine axis number

t = TOA unit number

There are further internal system data blocks with identifier.

Reactions:

- Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:

Correct the machine data or undo the changes made.

Please inform the authorized personnel/service department. There are two determining machine data for cycle programs:

- $MN_MM_NUM_MAX_FUNC_NAMES = max. number of all cycle programs, error number = 2 shows that this value is too small.
- $MN_MM_NUM_MAX_FUNC_PARAM = max. number of all parameters defined in the cycle programs, error number = 2 shows that this value is too small

(If these MDs are modified, the memory backup is retained)

The following applies to macro definitions:

- $MN_MM_NUM_USER_MACROS = max. number of all macro definitions, error number = 2 shows that this value is too small.

(If these MDs are modified, the memory backup is retained)

The following applies to GUD variables:

- $MN_MM_NUM_GUD_MODULES = max. number of GUD data blocks per area (NCK/channel) (if GD1, GD2, GD3, GD9 are to be defined, then the value must be = 9 and not e.g. = 4).
- $MN_MM_NUM_GUD_NAMES_NCK = max. number of all NCK global GUD variables, error number = 2 shows that this value is too small.

© Siemens AG, 2002. All rights reserved

- $MN_MM_NUM_GUD_NAMES_CHAN = max. number of all channel-specific GUD variables in the channel, error number = 2 shows that this value is too small.
- $MN_MM_GUD_VALUES_MEM = total value memory of all GUD variables together, error number = 1 shows that this value is too small.

Program Continuation: Switch control OFF - ON.

6020 Machine data have been altered - now memory is reorganized

Definitions: Machine data have been changed that define the NC user memory allocation. Data management has restructured the memory in accordance with the altered machine data.

Reactions: - Alarm display.

Remedy: No remedial measures are required. Any user data that are required must be input again.

Program Continuation: Clear alarm with the RESET key. Restart part program

6030 Limit of user memory has been adapted

Definitions: Data management checks during power-up the actually available physical user memory (DRAM, DPRAM and SRAM) with the values in the system-specific machine data 18210 MM_USER_MEM_DYNAMIC, MD 18220 MM_USER_MEM_DPR and MD 18230 MM_-USERMEM_BUFFERED.

Reactions: - Alarm display.

Remedy: No remedial measures are required. The new maximum permissible value can be read from the reduced machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

6035 Instead of %1 KB the system has only %2 KB of free user memory of type "%3"

Parameters: %1 = Free memory capacity in KB defined for the control model

%2 = Actual maximum capacity of free memory in KB

%3 = Type of memory, "D" =non-battery-backed, "S" =battery-backed

Definitions: The alarm can only occur after a 'cold start' (=NCK start-up with standard machine data). The alarm is only a notice. There is no interference with any NCK functions. It shows that the NCK has less free user memory available than specified by Siemens for this control variant. The value of the actually available free user memory can also be taken from the machine data $MN_INFO_FREE_MEM_DYNAMIC, $MN_INFO_FREE_MEM_STATIC. Siemens supplies NCK with default settings that, depending on the model, have certain (free) memory space available for the specific settings of the actual applications. The original factory setting of NCK systems is thus that the alarm does not occur with a cold start.

Reactions: - Alarm display.

Remedy: Reasons for the message:

- The NCK contains compile cycle software, that uses so much memory space that the hardware cannot provide the required memory.
- The NCK runs on hardware that is not intended for this NCK release (i.e. that has not enough memory capacity).
- If the application runs properly with the remaining free user memory (i.e. can be started up without any errors), the message can simply be ignored.
- If the actual application cannot be configured because there is not enough memory capacity available, either the existing compile cycle must be reduced or, if possible, the system must be upgraded with additional memory space.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

6100 Error while creating %1, error number %2 %3

Parameters: %1 = Symbol name
%2 = Error code
%3 = If required, internal error identifier

Definitions: An error was detected while creating a compile cycle machine data. The error number specifies the type of error.
- Error number 1: Insufficient memory available
- Error number 2: Symbol in the NCK already exists
- Error number 3: Maximum possible number of symbols exceeded
- Error number 4: Invalid name prefix
- Error number 5: Illegal array size

Note: Other errors of this type could have occurred, but have not been displayed.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Error number 1: The memory reserved by machine data 12238 $MN_MM_CC_MD_MEM_SIZE has to be increased. If the error occurs while loading an archive, then the machine data must be increased "manually". To do this, either Edit the archive with 'arcedit' or Overwrite the MD in the MD picture and prevent the deletion of the machine data when writing the archive (MMC: Set Ask_for_CFG_RESET.INI = 1 in 'dino.ini'). Also refer to: Upgrade instructions P6.x.
- Error number 2: Error in the combination or while reloading compile cycles: Do not activate compile cycle.
- Error number 3: Error in the combination or while reloading compile cycles: Do not activate compile cycle.
- Error number 4: Error in the compile cycle: Do not activate compile cycle.
- Error number 5: Error in the compile cycle: Do not activate compile cycle.

Program Continuation: Switch control OFF - ON.

6401 Channel %1 tool change not possible: Empty location for tool %2 Duplo no. %3 on magazine %4 not available.

Parameters: %1 = Channel ID
%2 = String (identifier)
%3 = Duplo number
%4 = Magazine number

Definitions: The tool cannot be moved into the selected tool magazine. There is no appropriate location for this tool. A suitable location is mainly determined by the status. The status must indicate that this location is free, not disabled, not reserved and not co-occupied by a tool that is too large. Furthermore, it is important that the type of tool matches the type of any magazine location that may be free. (If, for example, all magazine locations are of the 'B' type and these are all free and the tool is of type 'A', then this tool cannot be put into this magazine).

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy:
- Check whether the magazine data have been defined correctly.
- Check whether there is still room in the magazine to add another tool; there may not be due to operating procedures.
- Check whether a location type hierarchy is defined and whether it, for example, does not allow insertion of a type 'A' tool in a free location with type 'B'.

Program Continuation: Clear alarm with the RESET key. Restart part program

6402 Channel %1 tool change not possible. Magazine no. %2 not available

Parameters: %1 = Channel ID
%2 = Magazine number

Definitions: The desired tool change is not possible. The magazine with the specified number is not available.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Check whether the magazine data have been defined correctly.
- Check whether the magazine is connected to the desired toolholder/spindle via a distance relation.

Program Continuation: Clear alarm with the RESET key. Restart part program

6403 Channel %1 tool change not possible. Magazine location number %2 on magazine %3 not available.

Parameters: %1 = Channel ID
%2 = Magazine number
%3 = Magazine location number

Definitions: The desired tool change is not possible. The specified magazine location is not contained in the specified magazine.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check whether the magazine data have been defined correctly.

Program Continuation: Clear alarm with the RESET key. Restart part program

6404 Channel %1 tool change not possible. Tool %2 not available or not usable

Parameters: %1 = Channel ID
%2 = String (identifier)

Definitions: The desired tool change is not possible. The specified tool does not exist or cannot be inserted.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Check whether the part program is written correctly.
- Check whether the tool data are correctly defined.
- Check whether there is a replacement tool which can be used for the specified tool.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

6405 Channel %1 command %2 has invalid PLC acknowledge parameter %3 - identifier %4

Parameters:
%1 = Channel ID
%2 = Command no.
%3 = PLC acknowledge parameter
%4 = Error code

Definitions:
The specified command has been answered by PLC with an invalid acknowledgement in the current combination. The following assignments are defined for "command no.":

1 Move tool, load or unload magazine

2 Prepare tool change

3 Execute tool change

4 Prepare tool change and execute with T command

5 Prepare tool change and execute with M command

7 Terminate canceled tool command

8 Check tool movement with reservation

9 Check tool movement

0 Transport acknowledgement

Parameters 2 and 3 designate the PLC command and the status number of the acknowledgement.

Example: Parameter 4 of the alarm message is 10. It is not defined whether a buffer location for asynchronous tool motion must be reserved. In the example, the parameter is ignored by the NCK. Further possible causes for the alarm: The tool change defined by the command is not possible. The magazine location specified in the invalid parameter does not exist in the magazine.

The 3rd parameter - error identification - gives a more detailed description of the alarm. Meanings:

- 0 = not defined
- 1 = status not allowed or undefined status received by PLC
- 2 = source and/or target magazine no./location no. unknown
- 3 = not defined
- 4 = target magazine no. and/or location no. in tool motion command not end target
- 5 = not defined
- 6 = source and/or target magazine no./location no. unknown during tool change
- 7 = PLC comm. with inconsistent data: either inconsistent magazine addresses in VDI or NCK command unequal to PLC acknowledgement or both
- 8 = PLC comm. with inconsistent data: while rejecting a tool, the tool to be rejected was unloaded asynchronously. NCK cannot perform a new selection.
- 9 = PLC comm. with inconsistent data: the command acknowledgement data wants to move a tool to a location that is occupied by another tool.
- 10 = it is not defined whether a buffer location for asynchronous tool motion must be reserved.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
Please inform the authorized personnel/service department. Erroneous PLC communication: Correct the PLC program.

Program Continuation:
Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

6406 Channel %1 PLC acknowledge for command %2 is missing

Parameters: %1 = Channel ID
%2 = Command no.

Definitions: There is still no acknowledgement from the PLC for the tool change. The NCK cannot continue processing until it receives this acknowledgement for the specified command number. Possible command number values are described for alarm 6405.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Erroneous PLC communication: Correct the PLC program.
- It is possible to release NCK with the PLC command 7 from the wait condition.

This aborts the waiting command.

Program Continuation: Clear alarm with the RESET key. Restart part program

6407 Channel %1 tool %2 cannot be placed in magazine %3 on location %4. Invalid definition of magazine!

Parameters: %1 = Channel ID
%2 = String (identifier)
%3 = Magazine number
%4 = Magazine location number

Definitions: A tool change request or a verification request was issued to put the tool in a location which does not satisfy the prerequisites for filling.
The following causes for the error are possible:
- Location is blocked or not free!
- Tool type does not match the location type!
- Tool possibly too large, adjacent locations are not free!

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Check whether the magazine data are correctly defined (especially the location type).
- Check whether the tool data are correctly defined (especially the location type).

Program Continuation: Clear alarm with the RESET key. Restart part program

6410 TO unit %1 tool %2 / Duplo no. %3 has reached its prewarning limit with D = %4

Parameters: %1 = TO unit
%2 = Tool identifier (name)
%3 = Duplo number
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring. The actual type of tool monitoring is a tool property (see $TC TP9). If replacement tools are not being used, the duplo number specified has no meaning. The alarm is triggered through the MMC or PLC (=OPI interface). The channel context is not defined. The TO unit was specified for this reason (see $MC_MM_LINK_TOA_UNIT).

Reactions:
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: For information only. The user must decide what to do.
Program Continuation: Clear alarm with the Delete key or NC START.

6411 Channel %1 tool %2 / Duplo no. %3 has reached its prewarning limit with D = %4

Parameters: %1 = Channel number
%2 = Tool identifier (name)
%3 = Duplo number
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring. The actual type of tool monitoring is a tool property (see $TC TP9).
If replacement tools are not being used, the duplo number specified has no meaning.
The alarm originates during NC program execution.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: For information only. The user must decide what to do.
Program Continuation: Clear alarm with the Delete key or NC START.

6412 TO unit %1 tool %2 / Duplo no. %3 has reached its monitoring limit with D = %4

Parameters: %1 = TO unit
%2 = Tool identifier (name)
%3 = Duplo number
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring.
The actual type of tool monitoring is a tool property (see $TC TP9).
If replacement tools are not being used, the duplo number specified has no meaning.
The alarm is triggered through the MMC or PLC (=OPI interface). The channel context is not defined. The TO unit was specified for this reason (see $MC_MM_LINK_TOA_UNIT).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: For information only. The user must decide what to do.
Program Continuation: Clear alarm with the Delete key or NC START.

6413 Channel %1 tool %2 / Duplo no. %3 has reached its monitoring limit with D = %4

Parameters: %1 = TO unit
%2 = Tool identifier (name)
%3 = Duplo number
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring.

© Siemens AG, 2002. All rights reserved

The actual type of tool monitoring is a tool property (see $TC TP9).
If replacement tools are not being used, the duplo number specified has no meaning.
The alarm originates during NC program execution.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: For information only. The user must decide what to do.

Program Continuation: Clear alarm with the Delete key or NC START.

6421

Channel %1 tool move not possible. Empty location for tool %2 Duplo no. %3 on magazine %4 not available.

Parameters: %1 = Channel ID
%2 = String (identifier)
%3 = Duplo number
%4 = Magazine number

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. The tool cannot be moved into the specified tool magazine. There is no appropriate location for this tool.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy:
- Check whether the magazine data have been defined correctly (e.g. the magazine must not be disabled).
- Check whether the tool data are correctly defined (for example, the tool location type must match the location types allowed in the magazine).
- Check whether there is still room in the magazine to add another tool; there may not be due to operating procedures.
- Check whether a location type hierarchy is defined and whether it, for example, does not allow insertion of a type 'A' tool in a free location with type 'B'.

Program Continuation: Clear alarm with the Delete key or NC START.

6422

Channel %1 tool move not possible. Magazine no. %2 not available.

Parameters: %1 = Channel ID
%2 = Magazine number

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. The magazine with the specified number is not available.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy:
- Check whether the magazine data have been defined correctly.
- If the PLC issued the command for motion: check whether the PLC program is correct.
- If the MMC issued the command for motion: check whether the MMC command was assigned correct parameters.

Program Continuation: Clear alarm with the Delete key or NC START.

6423

Channel %1 tool move not possible. Location %2 on magazine %3 not available.

Parameters: %1 = Channel ID
%2 = Magazine location number
%3 = Magazine number

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. The specified magazine location is not contained in the specified magazine.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Check whether the magazine data have been defined correctly.

Program Continuation: Clear alarm with the Delete key or NC START.

6424 Channel %1 tool move not possible. Tool %2 not available/not usable.

Parameters: %1 = Channel ID
%2 = String (identifier)

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. The status of the named tool does not allow movement of the tool. The named tool is not defined or not permitted for the command.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy:
- Check whether the tool status 'is being changed' ('H20') is set. If yes, then the appropriate tool change command must first be completed by the PLC. Then the tool should be able to be moved.
- Check whether the tool data are correctly defined. Has the correct T number been specified?
- Check whether the move command has been correctly parameterized. Is the desired tool at the source location? Is the target location suitable for taking the tool?
- Check whether the tool has already been loaded (if the alarm occurs while loading the tool).

Program Continuation: Clear alarm with the Delete key or NC START.

6425 Channel %1 tool %2 cannot be placed in magazine %3 on location %4. Invalid definition of magazine!

Parameters: %1 = Channel ID
%2 = String (identifier)
%3 = Magazine number
%4 = Magazine location number

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. A movement request was issued to put the tool in a location which does not satisfy the prerequisites for filling.
The following causes for the error are possible:
- Location is blocked or not free!
- Tool type does not match the location type!
- Tool possibly too large, adjacent locations are not free!
- If a tool is to be loaded or unloaded, the load/unload position must be of 'load location' type.
- If a tool is to be loaded or unloaded, is the magazine in question linked to the load/unload location?

See $TC_MDP1, $TC_MDP2.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy:
- Check whether the magazine data have been defined correctly.
- Check whether there is still room in the magazine to add another tool; there may not be due to operating procedures.
- Check whether a location type hierarchy is defined and whether it, for example, does not allow insertion of a type 'A' tool in a free location with type 'B'.

© Siemens AG, 2002. All rights reserved

- Check whether the magazine in question is linked to the load/unload location or whether a distance has been defined.
- Check whether the load/unload position is of 'load location' type.

See also $TC_MPP1.

Program Continuation: Clear alarm with the Delete key or NC START.

6430 Workpiece counter: overflow in table of monitored cutting edges.

Definitions: No more cutting edges can be entered in the piece counter table. As many cutting edges can be noted for the workpiece counter as are possible in total in the NCK. This means that if for each tool each cutting edge in each TO unit is used precisely once for a workpiece then the limit is reached.

If several workpieces are made on several toolholders/spindles simultaneously, it is possible to note 18100 MM_NUM_CUTTING_EDGES_IN_TOA cutting edges for the workpiece counter for all of the workpieces.

If this alarm occurs, it means that cutting edges used subsequently are no longer quantity monitored until the table has been emptied again, e.g. by means of the NC language command SETPIECE or by the relevant job from MMC, PLC (PI service).

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy:
- Was decrementing of the piece counter forgotten? Then program SETPIECE in the part program, or add the correct command in the PLC program.
- If the part program/PLC program is correct, then more memory should be set for tool cutting edges via the machine data $MN_MM_NUM_CUTTING_EDGES_IN_TOA (can only be performed with the necessary access rights!).

Program Continuation: Clear alarm with the Delete key or NC START.

6431 Function not allowed. Tool management/monitoring is not active.

Definitions: Occurs when a data management function is called which is not available because TOOLMAN is deactivated. For example, the language commands GETT, SETPIECE, GETSELT, NEWT, DELT.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Please inform the authorized personnel/service department.
- Make sure of how the NC is supposed to be configured! Is tool management or tool monitoring needed but not activated?
- Are you using a part program that is meant for a numerical control with tool management/tool monitoring? It is not possible to start this program on the numerical control without tool management/tool monitoring. Either run the part program on the appropriate NC control or edit the part program.
- Activate tool management/tool monitoring by setting the appropriate machine data. See $MN_MM_TOOL_MANAGEMENT_MASK, $MC_TOOL_MANAGEMENT_MASK
- Check whether the required option is set accordingly.

Program Continuation: Clear alarm with the Delete key or NC START.

6432 Function not executable. No tool assigned to tool holder/spindle

Parameters: %1 = Channel ID

Definitions: When an attempt is made to perform an operation that requires a tool to be located on the spindle. This can be the quantity monitoring function, for example.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Select another function, another toolholder/spindle, position tool on toolholder/spindle.

Program Continuation: Clear alarm with the Delete key or NC START.

6433 Channel %1 block %2 %3 not available with tool management

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: The symbol variable specified in %3 is not available with active tool management. The function GELSELT should be used with $P_TOOLP.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify program. If $P_TOOLP has been programmed, the GETSELT function should be used instead.

Program Continuation: Clear alarm with the RESET key. Restart part program

6434 Channel %1 block %2 NC command SETMTH not allowed because tool holder function not active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No master toolholder has been defined for the initial state ($MC_TOOL_MANAGEMENT_TOOLHOLDER = 0), therefore no toolholder is available. The NC command SETMTH has neither been defined. In this setting, the tool change is carried out referring to the master spindle. The master spindle is set with SETMS.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program (delete or replace SETMHT) or enable toolholder function via machine data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6441 Writing of $P_USEKT not allowed.

Definitions: An attempt was made to write the value of $P_USEKT. This is not possible since programming T= 'location number' with automatic setting of $P_USEKT is active.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Make sure of how the NC is supposed to be configured! (see bit16 and bit22 in $MC_TOOL_MANAGEMENT_MASK)
- Tool change with "Reject tool" is configured. If you now try to start this program on NC control with T='location number' with automatic setting of $P_USEKT this will not be possible.
- Either run the part program on the appropriate NC control or edit the part program.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

6442

Channel %1 function not executable. No tool assigned to desired magazine/magazine location %2.

Parameters: %1 = Channel ID

%2 = Magazine/magazine location no.

Definitions: PLC logic is presumably incorrect. Tool change with reject tool is configured. Preparatory command is pending. Selected tool is (e.g. from PLC) unloaded from its location. PLC acknowledges preparatory command with 'Repeat tool selection' (e.g. status =7). NCK cannot find the tool at the magazine location specified in the PLC command.

Or: Illegal operator intervention in an active tool selection (unloading of the tool to be selected) has occurred. Therefore the PLC acknowledgement fails.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: PLC programmer must note the following:

- Ensure that the tool is not removed from the specified magazine location (e.g. incorrect PLC program).
- Do not remove the tool from the programmed tool change before the final acknowledgement of the command (= unload).

!! It is however permissible to change the location of the tool to be loaded. The NCK can deal with this situation.

This alarm supplements Alarm 6405, if it contains the identifier 8. Therefore, the diagnosis should be easier.

Program Continuation: Clear alarm with the Delete key or NC START.

6450

Channel %1 tool change not possible. Invalid magazine location no. %2 in buffer magazine

Parameters: %1 = Channel ID

%2 = Magazine location number

Definitions: The desired tool change is not possible. The specified magazine location is either toolholder/spindle or empty.

Only the numbers of the buffer that are not toolholder/spindle may be programmed with the NC command TCI, i.e. the location number of a gripper is allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:

- Check whether the magazine data ($TC_MPP1) have been defined correctly.
- Check whether the alarm-causing program command – e.g. TCI – has been programmed correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6451

Channel %1 tool change not possible. No buffer magazine defined.

Parameters: %1 = Channel ID

Definitions: The desired tool change is not possible. No buffer magazine defined.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check whether the magazine data have been defined correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

6452

Channel %1 tool change not possible. Tool holder/spindle number = %2 not defined.

Parameters: %1 = Channel ID
%2 = Tool holder/spindle number

Definitions: The desired tool change is not possible. The toolholder/spindle number has not been defined.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check whether the toolholder number/spindle number and the magazine data have been defined correctly. (See system parameters $TC_MPP1, $TC_MPP5 of the buffer magazine)

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6453

Channel %1 tool change not possible. No assignment between toolholder/spindle no. = %2 and buffer magazine location %3

Parameters: %1 = Channel ID
%2 = Spindle no.
%3 = Location no.

Definitions: The desired tool change is not possible. No relation has been defined between the toolholder/spindle number and the buffer magazine location (Location No.)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Check whether the magazine data ($TC_MLSR) have been defined correctly.
- Check whether the alarm-causing program command – e.g. TCI – has been programmed correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6454

Channel %1 tool change not possible. No distance relation available.

Parameters: %1 = Channel ID

Definitions: The desired tool change is not possible. Neither the spindle nor the buffer magazine location have a distance relation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Check whether the magazine data ($TC_MDP2) have been defined correctly.
- Check whether the alarm-causing program command – e.g. TCI – has been programmed correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6500

NC memory full

Definitions: This alarm may occur when writing the file cc_resu.mpf if the buffered memory available is not sufficient. Note: During initial start-up, this can concern files from the NC file system, e.g. drive data, MMC files, FIFO files, NC programs, ...

Reactions: - Alarm display.

Remedy: Adjust the size of the buffered memory ($MN_MM_USER_MEM_BUFFERED) or increase the space available in the buffered memory, e.g. by unloading part programs that are no longer being used. Or decrease the size of the ring buffer (see $MC_RESU_RING_BUFFER_SIZE).

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

6510 Too many part programs in the NC memory

Definitions: The number of files in the file system (part of the NC memory) of the NC has reached the maximum number possible. Note: During initial start-up, this can concern files from the NC file system, e.g. drive data, MMC files, FIFO files, NC programs, ...

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or unload files (e.g. part programs), or
- Increase $MM_NUM_FILES_IN_FILESYSTEM.

Program Continuation: Clear alarm with the Delete key or NC START.

6520 The value of the machine data %1%2 is too low

Parameters: %1 = String: MD identifier
%2 = If required, field index

Definitions: The machine data $MN_MM_PROTOC_NUM_FILES specifies the number of protocol files for the protocol users. However, more types are used than configured.

Reactions: - Alarm display.

Remedy: Increase machine data $MN_MM_PROTOC_NUM_FILES.

Program Continuation: Clear alarm with the Delete key or NC START.

6530 Too many files in directory

Definitions: The number of files in one directory of the NCK has reached the maximum limit.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or unload files (e.g. part programs) in the respective directory, or
- Increase $MM_NUM_FILES_PER_DIR.

Program Continuation: Clear alarm with the Delete key or NC START.

6540 Too many directories in the NC memory

Definitions: The number of directories in the file system of the NCK has reached the maximum limit.

Reactions: - Alarm display.

Remedy:
- Delete or unload directory (e.g. workpiece), or
- Increase $MM_NUM_DIR_IN_FILESYSTEM.

Program Continuation: Clear alarm with the Delete key or NC START.

6550 Too many subdirectories

Definitions: The number of subdirectories in a directory of the NCK has reached the maximum limit.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or empty subdirectories in the respective directory, or
- Increase $MM_NUM_SUBDIR_PER_DIR.

Program Continuation: Clear alarm with the Delete key or NC START.

6560 Data format not allowed

Definitions: An attempt was made to write impermissible data in an NCK file. This error can occur in particular when the attempt was made to load binary data in the NCK as ASCII file.
The error can also occur during preprocessing of cycles (see $MN_PREPROCESSING_LEVEL) if the NC block is very long. In this case, subdivide the NC block.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
Remedy: Specify that the file concerned is a binary file (e.g. extension: .BIN).
Program Continuation: Clear alarm with the Delete key or NC START.

6570 NC memory full

Definitions: The NC card file system of the NCK is full. The task cannot be executed. Too many system files were created in the DRAM.
Reactions: - Alarm display.
Remedy: Start fewer "execute from external" processes.
Program Continuation: Clear alarm with the Delete key or NC START.

6580 NC memory full

Definitions: The NC card file system of the NCK is full. The task cannot be executed. To many files have been loaded
Reactions: - Alarm display.
Remedy: Delete or empty files (e.g. part programs).
Program Continuation: Clear alarm with the Delete key or NC START.

6600 NC card memory is full

Definitions: The NC card file system of the NCK is full. No more data can be stored on the NC card.
Reactions: - Alarm display.
Remedy: Delete the data on the PCMCIA card.
Program Continuation: Clear alarm with the Delete key or NC START.

6610 Too many files open on NC card

Definitions: Too many files are being accessed simultaneously on the NC card.
Reactions: - Alarm display.
Remedy: Repeat the action later.
Program Continuation: Clear alarm with the Delete key or NC START.

6620 NC card has incorrect format

Definitions: The NC card cannot be accessed because the format is incorrect.
Reactions: - Alarm display.
Remedy: Replace the NC card.
Program Continuation: Clear alarm with the Delete key or NC START.

6630 NC card hardware is defective

Definitions: The NC card cannot be accessed because the card is defective.
Reactions: - Alarm display.
Remedy: Replace the PCMCIA card.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

6640 NC card is not inserted

Definitions: The NC card cannot be accessed because the card is not plugged in.
Reactions: - Alarm display.
Remedy: Plug in the NC card.
Program Continuation: Clear alarm with the Delete key or NC START.

6650 Write protection of NC card is active

Definitions: The NC card cannot be accessed because the write protection is active.
Reactions: - Alarm display.
Remedy: Deactivate the write protection.
Program Continuation: Clear alarm with the Delete key or NC START.

6660 'Flash File System' option is not set

Definitions: The NC card cannot be accessed because the option is not enabled.
Reactions: - Alarm display.
Remedy: Buy option.
Program Continuation: Clear alarm with the Delete key or NC START.

6670 NC card read active

Definitions: The alarm is active while the contents of the NC card are being read out. The FFS cannot be accessed during this period.
Reactions: - Alarm display.
Remedy: Wait until the read-out procedure is terminated.
Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

6671 NC card write active

Definitions: The alarm is active while the contents of the NC card are being written. The FFS cannot be accessed during this period. If the power is switched off while the alarm is active, the contents of the PCMCIA card are destroyed!
Reactions: - Alarm display.
Remedy: Wait until the write procedure is terminated.
Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

6690 Cycles from NC card cannot be copied to the passive file system.

Definitions: There is not enough space in the file system that the directories specified in the $PCMCIA_FUNKTION_MASK can be copied from the NC card to the passive file system.
Reactions: - Alarm display.
Remedy: Delete data in the file system.
Program Continuation: Clear alarm with the Delete key or NC START.

6691 Cycles from the passive file system cannot be saved on the NC card

Definitions: There is not enough space on the NC card that the directories specified in the $PCMCIA_FUNKTION_MASK can be saved. It is possible that cycles are lost during the next booting.
Reactions: - Alarm display.
Remedy: Delete data on the NC card or delete cycles not required.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

6692 Cycle %1 lost

Parameters: %1 = Name of cycle

Definitions: A cycle has been changed and due to a power failure, the backup on the PC card could not be terminated properly. The cycle is lost.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.

Remedy: Import the cycle again.

Program Continuation: Switch control OFF - ON.

6693 File %1 lost

Parameters: %1 = Name of file

Definitions: Due to a power failure, a file change could not be terminated properly. The file is lost.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.

Remedy: Import the file again.

Program Continuation: Switch control OFF - ON.

6698 Unknown NC card (%1/%2). Writing not possible.

Parameters: %1 = actManufactorCode (manufacturer code read by the card)

%2 = actDeviceCode (memory code read by the card)

Definitions: The NC card cannot be accessed because a valid write algorithm is not available for the flash memory.

Reactions: - Alarm display.

Remedy: Use a compatible NC card or enter the new manufacturer code/device code in MD $MN_PERMISSIVE_FLASH_TAB after consultation with SIEMENS.

Program Continuation: Clear alarm with the Delete key or NC START.

6700 Channel %1 value of the machine data %2%3 is too low

Parameters: %1 = Channel number

%2 = MD identifier

%3 = If required, field index

Definitions: The machine data $MC_MM_PROTOC_NUM_ETP_STD_TYP specifies the number of default event types for the protocol users. However, more types are used than configured.

Reactions: - Alarm display.

Remedy: Increase machine data $MC_MM_PROTOC_NUM_ETP_STD_TYP.

Program Continuation: Clear alarm with the Delete key or NC START.

7000 Too many compile cycle alarms defined

Definitions: Too many alarms are defined for the compile cycles. On powering up, the quantity was exceeded when defining a new CC alarm.

Reactions: - Alarm display.

Remedy: Apart from reducing the number of CC alarms, no remedial measures are possible at the present time. (contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.: see alarm 1000)

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

7010 Range of MMC alarm numbers for compile cycles exceeded

Definitions: A fixed quantity of alarm numbers (100) is reserved for the compile cycles. This has been exceeded when defining a new CC alarm. (The valid range is between 0 and 4999).

Reactions: - Alarm display.

Remedy: Define the CC alarm numbers in the valid range from 0 to 4999.

Program Continuation: Clear alarm with the Delete key or NC START.

7020 Compile cycle alarm number has not been defined

Definitions: The alarm ID used by the manufacturer is not known to the system. This was not allocated when the alarms were generated.

Reactions: - Alarm display.

Remedy: The alarm can have 2 possible causes:

- The alarm number has not been defined. A definition must still be made.
- The call parameter used is not the same as the one transferred by the NCK.

Program Continuation: Clear alarm with the Delete key or NC START.

7100 Compile cycles VDI area: %1 byte for inputs and %2 byte for outputs. Maximum %3 bytes available.

Parameters: %1 = String (machine data)
%2 = String (machine data)
%3 = Max. length for interface

Definitions: The sum of the input and output bytes at the VDI user interface for the compile cycles exceeds the maximum quantity of 400 bytes.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Set the machine data for dividing up the VDI user interface of the compile cycles (DB 9) into input and output bytes in accordance with the functions in the compile cycles. The maximum quantity of 400 bytes must not be exceeded. There are no restrictions concerning the division into input and output bytes.

Program Continuation: Switch control OFF - ON.

7200 Problem with externally linked compile cycle %1 %2

Parameters: %1 = Internal number
%2 = could be

Definitions: Problem with externally linked compile cycles.

Reactions: - Alarm display.

Remedy: See function description of the compile cycle!

Program Continuation: Clear alarm with the Delete key or NC START.

7201 Assertion error in %1 line %2

Parameters: %1 = String (path with program name)
%2 = String (line number)

Definitions: This alarm is purely a development alarm. It only occurs with externally linked compile cycles.

© Siemens AG, 2002. All rights reserved

Reactions: - The NC switches to follow-up mode.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Alarm reaction delay is canceled.

Remedy: Consultation with CC developer

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

7202 Missing option bit for %1: %2 <hex>

Parameters: %1 = (string) name of the specific .elf file
%2 = (int) required option bit (hex)

Definitions: Alarm for SIEMENS compile cycles. This alarm appears when the option bit required for a SIEMENS compile cycle is not set.

Reactions: - NC not ready.
- Channel not ready.
- Interface signals are set.
- Alarm display.
- Alarm reaction delay is canceled.

Remedy: Set the required option bit or delete the .elf file from the Flash File System.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

8000 Channel %1 option 'user interrupt programs' not set

Parameters: %1 = Channel number

Definitions: The input signals of NCK inputs are required in order to activate the interrupt routines and rapid lift from contour. This function is not included in the basic version and must be retrofitted when needed.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Do not use rapid interrupt inputs or contact the machine manufacturer with a view to retrofitting this option!

Program Continuation: Clear alarm with the RESET key. Restart part program

8010 Option 'activation of more than %1 axes' not set

Parameters: %1 = Number of axes

Definitions: More machine axes have been defined through the channel-specific MD 20070 AXCONF_MACHAX_USED than are allowed in the system.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.
- NC not ready.
- Mode group not ready, also effective for single axes

Remedy: Please inform the authorized personnel/service department. The sum of all axes that have been configured through the channel-specific MD 20070 AXCONF_MACHAX_USED, must not exceed the maximum number of axes (dependent on configuration -> option, basic version: 4 axes).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

8020 Option 'activation of more than %1 channels' not set

Parameters: %1 = Number of channels

Definitions: A 2nd channel has been indicated but the corresponding option does not exist.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: In the system-specific MD 10010 ASSIGN_CHAN_TO_MODE_GROUP, reduce the number of channels to 1 or retrofit the option for a 2nd channel.

Program Continuation: Switch control OFF - ON.

8021 Option 'activation of more than %1 mode groups' not set

Parameters: %1 = Number of mode groups

Definitions: The option for the number of mode groups is not compatible with the activated mode group.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Add option for more mode groups. Activate fewer mode groups.

Program Continuation: Switch control OFF - ON.

8022 Option 'activation of more than %1KB SRAM' not set

Parameters: %1 = Memory size

Definitions: The option for memory extension does not correspond to the active SRAM.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Buy option
- Activate less SRAM

Program Continuation: Switch control OFF - ON.

8030 Channel %1 block %2 option 'interpolation of more than 4 axes' not set

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The option for the number of interpolating axes does not correspond to the number of axes programmed in the interpolation group.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Option: "Interpolation of more than 4 axes" (the number of axes that is then allowed can be set in this option) or specify in the part program as many (or fewer, as required) axes corresponding to the configuration of the control.

Program Continuation: Clear alarm with the RESET key. Restart part program

8032 Option 'activation of more than %1 link axes' not set

Parameters: %1 = Number of axes

Definitions: The option for the number of link axes does not match the number of axes programmed in MD $MN_AXCONF_LOGIC_MACHAX_TAB.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: • Buy option
• Configure fewer link axes

Program Continuation: Clear alarm with the RESET key. Restart part program

8034 Option 'activation of axis containers' not set

Definitions: The option for activating the axis container function in MD $MN_AXCONF_LOGIC_MACHAX_TAB is not enabled.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: • Buy option
• Do not configure any containers

Program Continuation: Clear alarm with the RESET key. Restart part program

8036 Option: it is not allowed to set different IPO cycles or position control cycles with NCU link.

Definitions: The option for activating the FAST_IPO_LINK has not been set. For NCU link, all Ipo or position control cycles must then be equal (see FAST-IPO-LINK description).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: • Buy option
• Do not activate different Ipo or position control cycles (see MN_IPO_SYSCLOCK_TIME_RATIO and MN_POSCTRL_SYSCLOCK_TIME_RATIO).

Program Continuation: Switch control OFF - ON.

8038 Option 'activation of more than %1 lead link axes' not set

Parameters: %1 = Number of axes

Definitions: The option for the number of lead link axes does not match the number of configured axes in the MD $MA_AXCONF_ASSIGN_MASTER_NCU.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: • Buy option
• Configure fewer lead link axes

Program Continuation: Clear alarm with the RESET key. Restart part program

8040 Machine data %1 reset, corresponding option is not set

Parameters: %1 = String: MD identifier

Definitions: A machine data has been set that is locked by an option.

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. For retrofitting the option, please refer to your machine manufacturer or to a sales representative of SIEMENS AG, A&D MC.

Program Continuation: Clear alarm with the Delete key or NC START.

8041 Axis %1: MD %2 reset, corresponding option not sufficient

Parameters: %1 = Axis number

%2 = String: MD identifier

Definitions: All of the axes selected in the machine data of the assigned option are used. Safety functions have been selected for too many axes in the axial machine data.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.
- Mode group not ready.
- Channel not ready.
- Channel not ready.

Remedy: -

Program Continuation: Switch control OFF - ON.

8044 Option for IPO cycle time %1 ms not set

Parameters: %1 = Impermissible IPO cycle time

Definitions: The option for activation of an IPO cycle time of %1 ms has not been set.

Option - Permiss. IPO cycle time:
- Option-free >= 8ms
- 1. 1st step >= 6ms
- 2. 2nd step >= 4ms
- 3. 3rd step >= 2ms
- 4. 4th step <2ms

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy:
- Buy option
- Increase IPO cycle time (e.g. via MD IPO_SYSCLOCK_TIME_RATIO)

Program Continuation: Switch control OFF - ON.

8045 Option for selected cycle settings not set

Definitions: The option for the 810D Powerline for activation of the same current/speed/position controller/IPO cycle time grid as with the 840D is not set. Without the option, only the set values of the 810D Standard are permitted.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy:
- Buy option
- Set (current/speed controller) cycle times to 810D default values.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

8080 %1 options are activated without setting the license key

Parameters: %1 = Number of non-licensed options
Definitions: An option was activated but no license key set to prove the purchase of the option.
Reactions: - Alarm display.
Remedy: Generate and enter license key via Internet.
Program Continuation: Clear alarm with the Delete key or NC START.

8081 %1 options are activated that are not licensed by the license key

Parameters: %1 = Number of non-licensed options
Definitions: Options were activated, that are not licensed by the license key entered.
Reactions: - Alarm display.
Remedy: Generate and enter license key via Internet.
Program Continuation: Clear alarm with the Delete key or NC START.

8082 The license key was entered three times, Power On required before next try.

Definitions: The license can be entered three times max (correctly or incorrectly).
Reactions: - Alarm display.
Remedy: Execute NCK Power On and enter the license key (correctly).
Program Continuation: Clear alarm with the Delete key or NC START.

8098 Invalid combination of options (%1)

Parameters: %1 = Bit mask of options
%1 = Bit 0 (LSB) Nibbling
%1 = Bit 1 External language
%1 = Bit 2 Neural quadrant error compensation
%1 = Bit 3 Measurement level 2
Definitions: The following restrictions apply to this module for the combination of options:
The option "2-channel" and the options "external language", "nibbling", "neural quadrant error compensation" and "measurement level 2" are mutually exclusive!
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.
- Mode group not ready.
- Channel not ready.
Remedy: Set the options accordingly.
Program Continuation: Switch control OFF - ON.

8100 Channel %1 block %2: function not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions:
- Impossible due to embargo regulations:
- 1. Synchronous actions: Writing of feed, override and axial offsets ($AA_VC, $AC_VC, $AA_OVR, $AA_VC and $AA_OFF) from synchronous actions as well as Continuous Dressing can be programmed only once in a block.
- 2. Extended measurement: 'Cyclic measurement' (MEAC) and 'Measurement from synchronous action' is not possible.

© Siemens AG, 2002. All rights reserved

• 3. Axis interpolation: The number of axes interpolating with one another must not exceed 4 (this also includes synchronous coupling of axes via synchronous actions "DO POS[X]=$A..." "DO FA[X]=$A...").

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

10203 Channel %1 NC start without reference point

Parameters: %1 = Channel number

Definitions: NC start has been activated in the MDA or AUTOMATIC mode and at least one axis that needs to be referenced has not reached its reference point.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Via the channel-specific MD 20700: REFP_NC_START_LOCK (NC Start without reference point) you can decide whether the axis must be referenced before NC Start or not. The start of referencing can be enabled channel-specific or axis-specific.

• Channel-specific reference point approach: The rising edge of the interface signal "activate referencing" (DB 21 - 28, DBX 1.0) starts an automatic sequence which starts the axes of the channel in the same sequence as specified in the axis-specific MD 34110 REFP_CYCLE_NR (axis sequence channel-specific referencing). 0: The axis does not participate in channel-specific referencing, but it must be referenced for NC Start, -1: The axis does not participate in channel-specific referencing, but it need not be referenced for NC Start, 1- 8: Starting sequence for the channel-specific referencing (simultaneous start at the same no.), 1 - 31: CPU type

• Axis-specific referencing: Press the direction key that corresponds to the approach direction in the axis-specific MD 34010 REFP_CAM_MDIR_IS_MINUS (reference point approach in minus direction).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10207 Channel %1 error when selecting or deselecting the digitize function

Parameters: %1 = Channel number

Definitions: An error has occurred on activating/deactivating the digitizing module; e.g. not in channel ready state, already activated, etc.

Reactions: - Alarm display.

Remedy: Press RESET.

Program Continuation: Clear alarm with the Delete key or NC START.

10208 Channel %1 continue program with NC start

Parameters: %1 = Channel number

Definitions: After block search with calculation, the control is in the desired state. The program can now be started with NC Start or the state can be changed for the time being with overstore/jog.

Reactions: - Alarm display.
- NC Stop on alarm.
- Program execution is aborted, not effective for single axes
- Interpreter stop

© Siemens AG, 2002. All rights reserved

Remedy: Press NC Start.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10209 Channel %1 internal NC stop after block search

Parameters: %1 = Channel number
Definitions: Internal alarm which initiates an NC Stop. The alarm is output if $MN_SEARCH_RUN_MODE ==1 and the last action block is activated after block search in the main run. Alarm 10208 is activated depending on the VDI signal PLC -> NCK channel DBB1.6.
Reactions: - NC Stop on alarm.
- Program execution is aborted, not effective for single axes
- Interpreter stop
Remedy: NC-Start
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10222 Channel %1 inter-channel communication not possible

Parameters: %1 = Channel number
Definitions: This channel has received a negative acknowledgment from the inter-channel communication because the destination channel number is not known, e.g.: START(x) or WAITE(x) but channel x has not been initialized
Reactions: - Alarm display.
Remedy: This is an indication of possible discrepancies. The program continues if no acknowledgment is called for.
Program Continuation: Clear alarm with the Delete key or NC START.

10223 Channel %1: Command %2 is already occupied

Parameters: %1 = Channel number
%2 = Event name
Definitions: This channel has received a negative acknowledgment from the inter-channel communication because this command is already active or has not yet been terminated, e.g.: INIT(x,"ncprog") but a program select request is already active for channel x.
Reactions: - Alarm display.
Remedy: This is an indication of possible discrepancies. The program continues if no acknowledgment is called for.
Program Continuation: Clear alarm with the Delete key or NC START.

10225 Channel %1: command denied

Parameters: %1 = Channel number
Definitions: The channel has received a command. The command cannot be executed.
Reactions: - Alarm display.
Remedy: Press RESET.
Program Continuation: Clear alarm with the Delete key or NC START.

10299 Channel %1 Auto-Repos function is not enabled

Parameters: %1 = Channel number
Definitions: The Auto-Repos function (operating mode) was selected in the channel but is not implemented.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: This message is purely informational.
Program Continuation: Clear alarm with the Delete key or NC START.

10600 Channel %1 block %2 auxiliary function during thread cutting active

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An auxiliary function output is programmed in a thread cutting block.
Reactions: - Alarm display.
Remedy: Consequential errors can occur if the machining path of the thread block is too short and further blocks (thread blocks) follow in which no machining stop may occur.
Possible remedial measures:
- Program a longer path and/or a lower traversing rate.
- Output auxiliary function in another block (program section).

Program Continuation: Clear alarm with the Delete key or NC START.

10601 Channel %1 block %2 zero velocity at block end point during thread cutting

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: This alarm occurs only when several blocks with G33 follow in succession. The block end velocity in the specified block is zero although a further velocity block follows. The reasons for this can be, for instance:
- G9
- Auxiliary function after motion
- Auxiliary function output before the motion of the following block
- Positioning axis in the block

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Modify the NC part program by removing any programmed "Stop at end of block" G09.
Modify general machine data 11110 AUXFU_GROUP_SPEC [n] for selecting the output time of an auxiliary function group by changing "Auxiliary function output before/after the movement" to "Auxiliary function output during the movement".
Bit 5 = 1: Auxiliary function output before movement
Bit 6 = 1: Auxiliary function output during movement
Bit 7 = 1: Auxiliary function output after movement

Program Continuation: Clear alarm with the RESET key. Restart part program

10604 Channel %1 block %2 thread lead increase too high

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The thread lead increase is causing an axis overload. A spindle override of 100% is assumed during verification.
Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Reduce the spindle speed, thread lead increase or path length in the NC program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10605 Channel %1 block %2 thread lead decrease too high

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The thread lead decrease is causing an axis standstill in the thread block.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Reduce the thread lead decrease or path length in the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10607 Channel %1 block %2 thread with frame not executable

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The current frame is corrupting the reference between the thread length and the thread lead.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm at block end.
- NC Start disable in this channel.

Remedy:
- Perform thread cutting with G33, G34, G35 without a frame.
- Use G63 or G331/G332.

Program Continuation: Clear alarm with the RESET key. Restart part program

10610 Channel %1 axis %2 not stopped

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An axis/spindle has been positioned over several NC blocks using the POSA/SPOSA instruction. The programmed target position had not yet been reached ("exact stop fine" window) when the axis/spindle was reprogrammed.

Example:

```
N100 POSA[U]=100
:
N125 X... Y... U... ; e.g.: U axis still travels from N100!
```

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check and correct the part program (analyze whether motion beyond block boundaries is appropriate here). Prevent block change by means of the keyword WAITP until the positioning axes have also reached their target position.

Example:

```
N100 POSA[U]=100
:
N125 WAITP[U]
N130 X... Y... U...
```

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

10620 Channel %1 block %3 axis %2 at software limit switch %4

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
%4 = String

Definitions: During the traversing motion, the system detected that the software limit switch would be crossed in the direction indicated. During block preparation, it was not yet possible to detect that the traversing range would be exceeded: Either there has been a motion overlay by the handwheel or a coordinate transformation is active.

Reactions: - Local alarm reaction.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm at block end.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Depending on the reason for this alarm being triggered, the following remedial measures should be undertaken:

- Handwheel override: Cancel the motion overlay and avoid this or keep it smaller when the program is repeated.
- Transformation: Check the preset/programmed zero offsets (current frame). If the values are correct, the tool holder (fixture) must be moved in order to avoid triggering the same alarm when the program is repeated, which would again cause the program to be aborted.

Program Continuation: Clear alarm with the RESET key. Restart part program

10621 Channel %1 axis %2 rests on software limit switch %3

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = String

Definitions: The specified axis is already stationary at the displayed software limit.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Machine data 36110 POS_LIMIT_PLUS/36130 POS_LIMIT_PLUS2 and 36100 POS_LIMIT_MINUS/36120 POS_LIMIT_MINUS2 must be checked for the software limit switches.
Shut down in JOG mode from the software limit switch.
Please inform the authorized personnel/service department.
Machine data:
Check the axis-specific interface signals: "2nd software limit switch plus" (DB 31 - 61, DBX 12.3) or "2nd software limit switch minus" (DB 31 - 61, DBX 12.2) check whether the 2nd software limit switch is selected.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10630 Channel %1 block %2 axis %3 at working area limit %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis, spindle number
%4 = String (+ or -)

Definitions: The specified axis violates the working area limitation. This is recognized only in the main run either because the minimum axis values could not be measured before the transformation or because there is a motion overlay.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm at block end.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- NC Start disable in this channel.

Remedy: Program other motion or do not perform overlaid motion.

Program Continuation: Clear alarm with the RESET key. Restart part program

10631 Channel %1 axis %2 rests at working area limit %3

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = String (+ or -)

Definitions: The specified axis reaches the working area limitation in JOG mode.

Reactions: - Alarm display.

Remedy: Setting data: Check 43420 WORKAREA_LIMIT_PLUS and 43430 WORKAREA_LIMIT_MINUS for the working area limitation.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10650 Channel %1 axis %2 incorrect gantry machine data, error code %3

Parameters: %1 = Channel number
%2 = Axis
%3 = Error no.

Definitions: An incorrect value was entered in the gantry-specific axial machine data. Further information can be derived from the error number.
- Error no. = 1 => either an incorrect gantry unit has been entered or the designation of the following axis is incorrect.
- Error no. = 2 => master axis has been specified more than once.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Correct the machine data:
MD 37100 GANTRY_AXIS_TYPE
0: No gantry axis 1: Master axis grouping 1 11: Slave axis grouping 1 2: Master axis grouping 2 12: Slave axis grouping 2 3: Master axis grouping 3 13: Slave axis grouping 3

Program Continuation: Switch control OFF - ON.

10651 Channel %1 illegal gantry configuration. Error code %2

Parameters: %1 = Channel number
%2 = Reason

Definitions: The gantry configuration set in the machine data is undefined. Gantry unit and reason for objection can be found in the transfer parameter.
The transfer parameter is made up as follows.
- %2 = error designation + gantry unit (XX).
- %2 = 10XX => no master axis declared
- %2 = 20XX => no slave axis declared

© Siemens AG, 2002. All rights reserved

- %2 = 3000 => different contents in MD 30550 slave axis and master axis
- %2 = 10000 => error: slave axis is geometry axis
- %2 = 11000 => error: competing position axis as slave axis
- %2 = 12000 => error: compile cycle axis as slave axis
- %2 = 13000 => error: gantry axis is spindle
- %2 = 14000 => error: gantry axis is Hirth geared

e.g. error no. 1001 = no master axis declared, grouping 1.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Correct the machine data:
MD 37100 GANTRY_AXIS_TYPE
0: No gantry axis
1: Master axis grouping 1
11: Slave axis grouping 1
2: Master axis grouping 2
12: Slave axis grouping 2
3: Master axis grouping 3
13: Slave axis grouping 3

Program Continuation: Switch control OFF - ON.

10652 Channel %1 axis %2 gantry warning threshold exceeded

Parameters: %1 = Channel number
%2 = Axis

Definitions: The gantry following axis has exceeded the warning limit specified in MD 37110 GANTRY_POS_TOL_WARNING.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.
1. Check axis (uneven mechanical movement?)
2. MD not set correctly (MD 37110 GANTRY_POS_TOL_WARNING). Changes to this MD take effect after a RESET.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10653 Channel %1 axis %2 gantry error threshold exceeded

Parameters: %1 = Channel number
%2 = Axis

Definitions: The gantry following axis has exceeded the error limit (actual value tolerance) specified in MD 37120 GANTRY_POS_TOL_ERROR.

Reactions: - NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
1. Check axis (uneven mechanical movement?)
2. MD not set correctly (MD 37120 GANTRY_POS_TOL_ERROR). A POWER ON is necessary after modifying the MD.
If the axes are not yet referenced, MD GANTRY_POS_TOL_REF is the trigger condition for the error message.
Program Continuation: Clear alarm with the RESET key. Restart part program

10654 Channel %1 waiting for synchronization start of gantry group %2

Parameters: %1 = Channel number
%2 = Gantry unit
Definitions: Alarm message appears when the axes are ready for synchronization. The gantry unit can now be synchronized. The actual value difference between master and slave axis is greater than the gantry warning threshold MD 37110 GANTRY_POS_TOL_WARNING. The synchronization must be started again explicitly with the start gantry synchronization interface signal (DB31-48, DBX 29.4).
Reactions: - Alarm display.
Remedy: Please inform the authorized personnel/service department. See Description of Functions (Special Functions), G1 Gantry Axis
Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10655 Channel %1 synchronization of gantry group %2 in progress

Parameters: %1 = Channel number
%2 = Gantry unit
Definitions: No further explanation.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10656 Channel %1 axis %2 gantry alarm not yet used

Parameters: %1 = Channel number
%2 = Axis
Definitions: No further explanation. Spare alarm, not used at the present time.
Reactions: - NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
Remedy: -
Program Continuation: Clear alarm with the RESET key. Restart part program

10700 Channel %1 block %2 NCK protection zone %3 violated during automatic or MDA mode

Parameters: %1 = Channel number
%2 = Block number
%3 = Protection zone number
Definitions: The workpiece-related NCK protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.

Remedy: Protection zone can be traversed after a new NC Start.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10701 Channel %1 block %2 channel-specific protection zone %3 violated during automatic or MDA mode

Parameters: %1 = Channel number
%2 = Block number
%3 = Protection zone number

Definitions: The workpiece-related channel-specific protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.

Remedy: Protection zone can be traversed after a new NC Start.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10702 Channel %1 NCK protection zone %2 violated during manual mode

Parameters: %1 = Channel number
%2 = Protection zone number

Definitions: The workpiece-related NCK protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

Remedy: Protection zone can be traversed after a new NC Start.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10703 Channel %1 channel-specific protection zone %2 violated during manual mode

Parameters: %1 = Channel number
%2 = Protection zone number

Definitions: The workpiece-related channel-specific protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

Remedy: Protection zone can be traversed after a new NC Start.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10704 Channel %1 block %2 protection zone monitoring is not guaranteed

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: New movements of a geometry axis which have been added could not be allowed for at the time of block preparation. It is therefore not certain that the protection zones will not be violated. This is just a warning message without further reactions.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Take other measures to ensure that the geometry axes motion, including the additional motion, does not violate the protection zones. (The warning comes nevertheless) or exclude additional motions.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10706 Channel %1 NCK protection zone %2 reached with axis %3 during manual mode

Parameters: %1 = Channel number
%2 = Protection zone number
%3 = Axis name

Definitions: The workpiece-related NCK protection zone has been reached with the specified axis. Note that another tool-related protection zone is still active. The workpiece-related protection zone can be traversed when the PLC has issued an enable signal.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

Remedy: Please inform the authorized personnel/service department. Protection zone can be traversed after enable signal from PLC.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10707 Channel %1 channel-specific protection zone %2 reached with axis %3 during manual mode

Parameters: %1 = Channel number
%2 = Protection zone number
%3 = Axis name

Definitions: The workpiece-related channel-specific protection zone has been reached with the specified axis. Note that another tool-related protection zone is still active. The workpiece-related protection zone can be traversed when the PLC has issued an enable signal.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

Remedy: Please inform the authorized personnel/service department. Protection zone can be traversed after enable signal from PLC.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

10710 Channel %1 block %2 conflict with centerless grinding

Parameters: %1 = Channel number
%2 = Spindle number

Definitions: Centerless grinding is active and a block has been processed that satisfies at least one of the following conditions:

- G96 active and regulating spindle is master spindle.
- Regulating spindle is in interdependent grouping.
- Axes of centerless transformation overlap with an active transformation and a tool is active.
- Constant wheel peripheral speed for the regulating spindle is active.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify program.

Program Continuation: Clear alarm with the RESET key. Restart part program

10720 Channel %1 block %3 axis %2 software limit switch %4

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
%4 = String (+ or -)

Definitions: For the axis, the programmed path violates the currently valid software limit switch. The alarm is activated when preparing the part program block.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Correct the NC program.
Please inform the authorized personnel/service department. Check the position of the axis as specified in the part program.
Machine data: 36100 POS_LIMIT_MINUS/36120 POS_LIMIT_MINUS2 and 36110 POS_LIMIT_PLUS/36130 POS_LIMIT_PLUS2 must be checked for the software limit switches.
Check the axis-specific interface signals: "2nd software limit switch plus/minus" (DB 31 - 61, DBX 12.2 and 12.3) to see whether the 2nd software limit switch is selected.
Check currently active zero offsets via the current frame.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10721 Channel %1 block %3 axis %2 software limit switch %4

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
%4 = String (+ or -)

Definitions: For the axis, the planned motion violates the software limit switch. The alarm is activated during the preparation of approach or rest blocks at REPOS.

Reactions: - Local alarm reaction.
- Interface signals are set.
- Alarm display.

Remedy: Check in the NC program and current positions.
Check the axis-specific interface signals "2nd software limit switch plus/minus" (DB31-61, DBX 12.2 or 12.3) to see whether the 2nd software limit switch is selected.
Check currently active zero offset via the current frame.
Check the machine data for the software limit switch (36100 POS_LIMIT_MINUS / 36120 POS_LIMIT_MINUS2 or 36110 POS_LIMIT_PLUS / 36130 POS_LIMIT_PLUS2).
Interrupt the NC program via NC reset.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

10730 Channel %1 block %3 axis %2 working area limitation %4

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
%4 = String (+ or -)

Definitions: This alarm is generated if it is determined during block preparation that the programmed path of the axis will result in exceeding the working area limitation.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy:
a) Check NC program for correct positional data and, if necessary, make corrections.
b) Check zero offsets (current frame)
c) Correct working area limitation via G25, or
d) Correct working area limitation via setting data, or
e) Deactivate working area limitation via setting data 43410 WORKAREA_MINUS_ENABLE=FALSE

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10731 Channel %1 block %3 axis %2 working area limitation %4

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
%4 = String (+ or -)

Definitions: For the axis, the planned motion violates the working area limit. The alarm is activated during the preparation of approach or rest blocks at REPOS.

Reactions:
- Local alarm reaction.
- Interface signals are set.
- Alarm display.

Remedy: Abort part program with reset.

Program Continuation: Clear alarm with the RESET key. Restart part program

10740 Channel %1 block %2 too many empty blocks in WAB programming

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: It is not allowed to program more blocks than specified by machine data MC_WAB_MAXNUM_DUMMY_BLOCKS between the WAB block and the block determining the approach and retraction tangent.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10741 Channel %1 block %2 direction reversal with WAB infeed motion

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: A safety distance which has been programmed is located perpendicular to the machining plane and not between the start and end point of the WAB contour.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10742 Channel %1 block %2 WAB distance invalid or not programmed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Possible causes:
- In a WAB block, the parameter DISR has not been specified or its value is less than or equal to 0.
- During approach or retraction with circle and active tool radius, the radius of the internally generated WAB contour is negative. The internally generated WAB contour is a circle with a radius which, when offset with the current offset radius (sum of tool radius and offset value OFFN), yields the tool center point path with the programmed radius DISR.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10743 Channel %1 block %2 WAB programmed several times

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: An attempt has been made to activate a WAB motion before a previously activated WAB motion was terminated.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10744 Channel %1 block %2 no valid WAB direction defined

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The tangent direction for smooth approach or retraction is not defined.

Possible causes:
- In the program, no block with travel information follows the approach block.
- Before a retraction block, no block with travel information has been programmed in a program.
- The tangent to be used for WAB motion is vertical to the current machining plane.

© Siemens AG, 2002. All rights reserved

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10745 Channel %1 block %2 WAB end position not clear

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the WAB block and in the following block, the position has been programmed perpendicular to the machining direction. In the WAB block, no position has been indicated in the machining plane.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program. Either remove the position data for the infeed axis from the WAB block or the following block, or program a position in the machining plane in the WAB block as well.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10746 Channel %1 block %2 block search stop for WAB

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A block search stop has been inserted between a WAB approach block and the following block defining the tangent direction or between a WAB approach block and the following block defining the end position.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10747 Channel %1 block %2 retraction direction not defined for WAB

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a WAB retraction block with quarter circle or semi-circle (G248 or G348), the end point in the machining plane was not programmed, and either G143 or G140 without tool radius compensation is active.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program. The following changes are possible:
- Indicate end point in the machining plane in the WAB block.
- Activate tool radius compensation (effective for G140 only, not for G143).
- State retraction side explicitly with G141 or G142.
- Perform retraction with a straight line instead of a circle.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10748 Channel %1 block %2 illegal retract plane with WAB

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: By means of DISRP a position of the retraction plane has been programmed which is not situated between the safety distance (DISCL) and the starting point (during approach) and/or end point (during retraction) of the WAB movement.

Reactions: - Correction block is reorganized.
- Local alarm reaction.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm at block end.

Remedy: Modify part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10750 Channel %1 block %2 tool radius compensation activated without tool number

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A tool T... must be selected so that the control can make allowance for the associated compensation values.
A correction data block (D1) containing the correction values (parameter P1 - P25) is automatically assigned to each tool (T number).
Up to 9 correction data blocks can be assigned to a tool by specifying the required data block with the D number (D1 - D9).
The cutter radius compensation (CRC) is allowed for if function G41 or G42 is programmed. The correction values are contained in parameter P6 (geometry value) and P15 (wear value) of the active correction data block Dx.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- Interpreter stop

Remedy: Before calling the CRC with G41/G42, program a tool number under the address T...

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10751 Channel %1 block %2 danger of collision due to tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The "Bottleneck detection" (calculation of intersection for the following compensated traversing blocks) has not been able to calculate a point of intersection for the reviewed number of traversing blocks. It is therefore possible that one of the equidistant paths violates the workpiece contour.

© Siemens AG, 2002. All rights reserved

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Please inform the authorized personnel/service department. Check the part program and modify the programming if possible such that inside corners with smaller paths than the correction value are avoided. (Outside corners are not critical because the equidistants are lengthened or intermediate blocks are inserted so that there is always a point of intersection).

Increase the number of reviewed traversing blocks via machine data 20240 CUTCOM_MAXNUM_CHECK_BLOCKS (default: 3), resulting in an increase in the extent of calculation and therefore also the block cycle time.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10752 Channel %1 block %2 overflow of local block buffer with tool radius compensation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The cutter radius compensation must buffer a variable number of intermediate blocks in order to enable calculation of the equidistant tool path for each NC block. The size of the buffer cannot be determined by simple means. It depends on the number of blocks without traversing information in the compensation plane, the number of contour elements to be inserted and the shape of the curvature in spline and polynomial interpolation.

The size of the buffer is fixed by the system and cannot be changed via the MDs.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Please inform the authorized personnel/service department. Reduce the size of the buffer that has been assigned by modifying the NC program. By avoiding:

- Blocks without traversing information in the compensation plane
- Blocks with contour elements having a variable curvature (e.g. ellipses) and with curvature radii that are smaller than the compensation radius. (Such blocks are divided up into several subblocks).

Reduce the number of reviewed blocks for collision monitoring (MD 20240 CUTCOM_MAXNUM_CHECK_BLOCKS).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10753 Channel %1 block %2 selection of the tool radius compensation only possible in linear block

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Selection of cutter radius compensation with G41/G42 may only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active.

In the block with G41/G42, at least one axis in the plane G17 to G19 must be written. It is always advisable to write both axes because, as a rule, both axes are traversed when selecting the compensation.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Correct the NC program and put the compensation selection in a block with linear interpolation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10754

Channel %1 block %2 deselection of the tool radius compensation only possible in linear block

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Deselection of cutter radius compensation with G40 can only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active.

In the block with G40, at least one axis in the plane G17 to G19 must be written. It is always advisable to write both axes because, as a rule, both axes are traversed when deselecting the compensation.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Correct the NC program and put the compensation selection in a block with linear interpolation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10755

Channel %1 block %2 selection of the tool radius compensation via KONT not possible at the current starting point

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: When activating the cutter radius compensation with KONT the starting point of the approach block is within the compensation circle and therefore already violates the contour.

If the cutter radius compensation is selected with G41/G42, the approach behavior (NORM or KONT) determines the compensation movement if the present actual position is behind the contour. With KONT, a circle is drawn with the cutter radius around the programmed initial point (= end point of the approach block). The tangent that passes through the current actual position and does not violate the contour is the approach movement.

If the start point is within the compensation circle around the target point, no tangent passes through this point.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Place selection of the CRC such that the starting point of the approach movements comes to rest outside of the correction circle around the target point (programmed traversing movements > compensation radius). The following possibilities are available:
- Selection in the previous block
- Insert intermediate block
- Select approach behavior NORM

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10756

Channel %1 block %2 deselection of the tool radius compensation via KONT not possible at the programmed end point

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: On deselection of the cutter radius compensation, the programmed end point is within the compensation circle. If this point were in fact to be approached without compensation, there would be a contour violation.

If the cutter radius compensation is deselected via G40, the approach behavior (NORM or KONT) determines the compensation movement if the programmed end point is behind the contour. With KONT, a circle is drawn with the cutter radius about the last point at which the compensation is still active. The tangent passing through the programmed end position and not violating the contour is the retraction movement.

If the start point is within the compensation circle around the target point, no tangent passes through this point.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Place deselection of the CRC such that the programmed end point comes to rest outside the compensation circle around the last active compensation point. The following possibilities are available:

- Deselection in the next block
- Insert intermediate block
- Select retract behavior NORM

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10757

Channel %1 block %2 changing the compensation plane while tool radius compensation is active not possible

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In order to change the compensation plane (G17, G18 or G19) it is first necessary to deselect the cutter radius compensation with G40.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Insert an intermediate block in the part program using the correction deselection. After the plane change, the cutter radius compensation is to be selected in an approach block with linear interpolation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10758

Channel %1 block %2 curvature radius with variable compensation value too small

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The current cutter radius compensation (the cutter used) is too large for the programmed path radius.

In a block with variable tool radius compensation, a compensation must be possible either anywhere or nowhere on the contour with the smallest and the largest compensation value from the programmed range. There must be no point on the contour in which the curvature radius is within the variable compensation range.

© Siemens AG, 2002. All rights reserved

If the compensation value varies its sign within a block, both sides of the contour are checked, otherwise only the compensation side.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Use smaller cutters or allow for a part of the cutter radius at the time of contour programming.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10759 Channel %1 block %2 path is parallel to tool orientation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a block with spline or polynomial interpolation, the corrected path runs in at least one point parallel to the tool orientation, i.e. the path has a tangent perpendicular to the compensation plane.

Straight lines running parallel to the tool orientation are permissible, as well as circles, with a circle plane that is perpendicular to the compensation plane (application in smooth retraction from a slot).

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Do not use splines or polynomials when writing the contour section, but straight lines and circles instead. Divide up the tool piece geometry and deselect the cutter radius compensation between the various sections.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10760 Channel %1 block %2 helical axis is not parallel to tool orientation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With active tool radius compensation a helix is only permissible if the helix axis is parallel to the tool, i.e. the circle plane and the compensation plane must be identical.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Orient helix axis perpendicular to the machining plane.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10761 Channel %1 block %2 tool radius compensation for ellipse with more than one revolution not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When machining the inside of an ellipse, in parts of the ellipse the curvature radii are greater than or smaller than the cutter radius compensation.

In ellipses, in this case the block would be split up into 4 subblocks with curvature radii that are greater than and less than the compensation radius.

© Siemens AG, 2002. All rights reserved

Over several revolutions, there would be a tremendous increase in the amount of calculation required by the unlimited number of resulting subblocks, and therefore this situation is rejected by the error message.

If compensation is possible everywhere or nowhere on the ellipse, then ellipses are also permissible that cover more than one full revolution.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Use cutter with smaller radius or program motion block on blocks with no more than one revolution.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10762 Channel %1 block %2 too many empty blocks between two traversing blocks with active tool radius compensation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The maximum permissible number of empty blocks is limited by a machine data.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy:
- Modify part program
- Modify machine data
- Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10763 Channel %1 block %2 path component of the block in the compensation plane becomes zero

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Due to the collision monitoring with active tool radius compensation, the path component of the block in the compensation plane becomes zero. If the original block contains no motion information perpendicular to the compensation plane, it means that this block is excluded.

Reactions:
- Alarm display.

Remedy:
- The behavior is correct at narrow locations that cannot be machined with the active tool.
- Modify the part program if necessary.
- Use tool with smaller radius if necessary.
- Program CDOF.

Program Continuation: Clear alarm with the Delete key or NC START.

10764 Channel %1 block %2 discontinuous path with active tool radius compensation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: This alarm occurs when, with active tool radius compensation, the starting point used for calculating the compensation is not identical to the end point of the preceding block.

© Siemens AG, 2002. All rights reserved

This situation can occur, for example, when a geometry axis is traversed between two positions as a positioning axis or when, with an active kinematic transformation (e.g. 5-axis transformation) the tool length compensation is altered.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10765 Channel %1 block %2 3D tool radius compensation not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm occurs when an attempt is made to activate the 3D tool radius compensation even though the option required for this is not fitted in the control.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Use another software version. The option cannot be activated by altering machine data because the necessary code is not physically available.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10766 Channel %1 illegal change of surface orientation between block %2 and block %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Block number, label

Definitions: This alarm occurs with 3D face milling when, at the time of block transition, the surface defined in the first block is continued in the second block with the rear side of the surface defined there. The block number in the alarm designates the second block.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10767 Channel %1 block %2 processing with tilt angle unequal 0 not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When face milling with a torus milling cutter, the tilt angle must be 0 if the surface normal vector and the tool orientation include an angle that is less than the limiting angle given by the machine data 21082 CUTCOM_PLANE_ORI_LIMIT, i.e. in this case only the lead angle may be unequal to 0.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program. If necessary, use another tool (ball end mill).
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10768 Channel %1 block %2 illegal tool orientation with 3D tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm can occur with 3D face milling: The angle between the surface normal vector of the surface to be machined and the extremal surface normal vector of the tool surface is smaller than the limit value given by the machine data 21080 CUTCOM_PARALLEL_ORI_LIMIT, or the tool is oriented such that machining would have to be performed from the rear side of the surface. In this case, the extremal surface normal vector is the vector whose direction deviates most from the direction in the tool point (i.e. parallel to the tool longitudinal axis).
With cylindrical tools or tools which end in a cylindrical part (e.g. the standard torus milling cutter), this vector is positioned perpendicular to the tool vector. For this type of tool, the alarm indicates that the angle between the tool longitudinal axis of, for example a side line of the cylinder, and the surface to be machined is smaller than the minimum permissible value. With tools whose (valid) surface ends in a conical part instead of a cylindrical part (e.g. a beveled cutter or a torus milling cutter where the torus is defined to be smaller than 90 degrees), this alarm indicates that the angle between a side line of the taper and the surface to be machined is smaller than the minimum permissible value.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program. If necessary, use another tool.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10769 Channel %1 block %2 Illegal surface normal vector with 3D tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In 3D face milling, surface normal vector and path tangent vector must theoretically be perpendicular to one another, i.e. they must be at 90° to one another. Since both vectors can be programmed independently of each other, deviations from this angle are possible and allowed. This alarm is generated when the angle between surface normal vector and path tangent vector becomes less than the limit angle given by the machine data 21084 CUTCOM_PLANENORMAL_PATH_LIMIT.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10770 Channel %1 block %2 change of corner type due to change of orientation with active tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The type of a corner (inside or outside corner) depends not only on the programmed path but also on the tool orientation. For this purpose, the programmed path is projected in the plane perpendicularly to the actual tool orientation and the corner type is determined there. If a change in orientation is programmed (in one or several blocks) between two traversing blocks, resulting in the type of corner at the end of the first traversing block being different from that at the start point of the second block, the above error message is issued.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10771 Channel %1 block %2 overflow of local block buffer due to orientation smoothing

Parameters:
%1 = Channel number
%2 = Block number, label

Definitions: This error occurs when more blocks must be buffered than memory space is available. This error can only occur when the software has been incorrectly configured.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Increase size of local buffer area.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10772 Channel %1 block %2 illegal orientation change when activating or deactivating 3D face cutting

Parameters:
%1 = Channel number
%2 = Block number, label

Definitions: In face milling, no intermediate blocks with pure orientation change are allowed between the activation block and the first correction block or between the last correction block and the deactivation block (3D tool radius compensation).

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10773 Channel %1 illegal tool orientation in block %2 at inside corner with block %3

Parameters:
%1 = Channel number
%2 = Block number, label
%3 = Block number, label

Definitions: On inside corners, the path of the traversing blocks concerned is reduced but the orientation change originally programmed in the block is retained and is now carried out in synchronism with the shortened path. Because of the ensuing changed relationship between path tangent, surface normal and tool orientation, singular points or points with impermissible side angle can occur in 3D face milling. This is not allowed.

© Siemens AG, 2002. All rights reserved

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10774 Channel %1 illegal tool dimensions with face cutting in block %2

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm occurs when illegal tool dimensions are programmed for face milling, e.g. negative tool radius, rounding radius zero or negative for tool types that require a rounding radius, taper angle zero or negative for tapered tools.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10775 Channel %1 illegal tool change with face cutting in block %2

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm occurs when a tool change has been programmed while 3D tool radius compensation is active with the result that the tool type changes or, if the tool type remains unchanged, at least one relevant tool dimension has changed as compared with the deselected tool. Depending on the tool type, relevant tool dimensions can be the tool diameter, the rounding radius or the taper angle. Changes to the tool length are allowed.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10776 Channel %1 block%2 axis %3 must be geometry axis if tool radius compensation is active

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name

Definitions: This alarm occurs when an axis that is required for tool radius compensation is not a geometry axis. With CUT2DF, the axis can be a positioning axis perpendicular to the machining plane; with all other types of compensation (CUT2DF, CUT3DC, CUT3DF, CUT3DFF), all geometry axes must be operated as such.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10777 Channel %1 block %2 tool radius compensation: too many blocks with suppression of compensation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The maximum permissible number of blocks with active compensation suppression with tool radius compensation is limited by the machine data CUTCOM_MAXNUM_SUPPR_BLOCKS.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy:
- Modify part program.
- Modify machine data.
- Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10778 Channel %1 block %2 preprocessing stop with active tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the setting data $SC_STOP_CUTCOM_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).
Reactions: - Alarm display.
- NC Stop on alarm at block end.

Remedy:
- Continue machining with CANCEL and Start.
- Modify part program.
- Set setting data $SC_STOP_CUTCOM_STOPRE to FALSE.

Program Continuation: Clear alarm with the Delete key or NC START.

10779 Channel %1 block %2 preprocessing stop with active tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the setting data $SC_STOP_CUTCOM_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).
To continue machining, activate the CANCEL key and perform a restart.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy:
- Continue machining with CANCEL and Start.
- Modify part program.
- Set setting data $SC_STOP_CUTCOM_STOPRE to FALSE.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10780 Channel %1 block %2 preprocessing stop with active tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the setting data $SC_STOP_CUTCOM_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).
To continue machining, activate the CANCEL key and perform a restart.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy:
- Continue machining with CANCEL and Start.
- Modify part program.
- Set setting data $SC_STOP_CUTCOM_STOPRE to FALSE.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10781 Channel %1 block %2 illegal orientation of involute with tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Tool radius compensation is possible for involutes only if the compensation plane matches the involute plane.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10782 Channel %1 block %2 illegal curve type with tool radius compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm occurs, if an attempt is made to apply the tool radius compensation to a curve type for which this function is not implemented. The only cause at present: Involute with 3D tool radius compensation.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10783 Channel %1 block %2 tool radius compensation type requires orientation transformation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm occurs, if an attempt is made to activate a tool radius compensation which must enable a tool orientation change and the “Orientation transformation” option is not available. This alarm can only occur if one of the following G code is active in the G code group 22:
- CUT3DC
- CUT3DCC
- CUT3DCCD

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy:
- Modify part program
- Install "Orientation transformation" option

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10784 Channel %1 block %2 illegal tool for tool radius compensation with constraint surface

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When activating the tool radius compensation with constraint surface, an illegal tool type is active.
Only cutting tools of the tool types 1 to 399 are admitted with the following exceptions:
- 111 ball end milling cutter
- 155 torus milling cutter
- 156 torus milling cutter
- 157 torus milling cutter

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Use another tool.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10790 Channel %1 block %2 plane change during linear programming with angles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The active plane was changed between the first and second subblock when programming two straight lines with angle parameters.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10791 Channel %1 block %2 invalid angle during linear programming

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No intermediate point was found when programming a contour consisting of two straight lines and an angle specification.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10792 Channel %1 block %2 illegal interpolation type during linear programming with angles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Only spline or linear interpolation is permitted for programming two straight lines with angle specification. Circular or polynomial interpolation is not allowed.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10793 Channel %1 block %2 second block missing during linear programming with angles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The second block is missing during programming of two straight lines with angle specification. This situation only occurs if the first subblock is also the last block of a program, or if the first subblock is followed by a block with a preprocessor stop.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10794 Channel %1 block %2 angle specification missing in second block during linear interpolation with angles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The angle is missing from the second block during programming of two straight lines with angle specification. This error can only occur if an angle was programmed in the preceding block, but no axis of the active plane was programmed in that block. The cause of the error may therefore also have been the intention to program a single straight line with an angle in the previous block. In this case, exactly one axis of the active plane must be programmed.

© Siemens AG, 2002. All rights reserved

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10795 Channel %1 block %2 end point specification during angle programming contradictory

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: During programming of a straight line, both positions of the active plane and an angle were specified (the position of the end point is over-specified), or the position of the programmed coordinate cannot be reached with the specified angle. If a contour consisting of two straight lines is to be programmed with angles, it is possible to specify the two axis positions of the plane and an angle in the second block. The error can also occur if, due to a programming error, the preceding block cannot be interpreted as the first subblock of such a contour. A block is interpreted as the first block of a two-block contour if an angle, but not an axis of the active plane, was programmed, and if the block is not already the second block of a contour.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10800 Channel %1 block %3 axis %2 is not a geometry axis

Parameters: %1 = Channel number

%2 = Axis name, spindle number

%3 = Block number, label

Definitions: With an active transformation or a frame with a rotation component the geometry axes are needed for block preparation. If a geometry axis has previously been traversed as positioning axis, it retains its status of "positioning axis" until it is again programmed as a geometry axis.

Because of the POSA motion beyond block boundaries, it is not possible to identify in the preprocessing run whether the axis has already reached its target position when the block is executed. This is, however, an unconditional requirement for calculating the ROT component of the frame or of the transformation.

If geometry axes are used as positioning axes, then:

1. No rotation may be specified in the current overall frame.

2. No transformation may be selected.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: After selecting transformation or frame, reprogram the geometry axis now operating as positioning axis (e.g. with WAITP) in order to revert the status to "geometry axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10805 Channel %1 block %2 repositioning after switch of geometry axes or transformation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the asynchronous subroutine the assignment of geometry axes to channel axes was changed or the active transformation modified.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

10810 Channel %1 block %2 master spindle not defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The function "Revolutional feedrate" (with G95 or G96), or "Rigid tapping" (with G331/ G332) has been programmed, although no master spindle is defined from which the speed could be derived. For the definition the MD 20090 SPIND_DEF_MASTER_SPIND is available for the default or the keyword SETMS in the part program, thus allowing each spindle of the channel to be redefined as master spindle.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Preset the master spindle with MD 20090 SPIND_DEF_MASTER_SPIND[n]=m (n ... channel index, m ... spindle no.) or define it with an identifier in an NC part program before a G function that requires a master spindle is programmed.

The machine axis that is to be operated as a spindle must be equipped in MD 35000 SPIND_ASSIGN_TO_MACHAX[n]=m (n ... machine axis index, m ... spindle no.) with a spindle number. Additionally, the MD 20070 AXCONF_MACHAX_USED[n]=m (n ... channel axis index, m ... machine axis index) must be used to assign it to a channel (channel axis index 1 or 2).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10820 Channel %1 rotary axis/spindle %2 not defined

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: Revolutional feed has been programmed for contouring and synchronous axes or for an axis/spindle. However, the rotary axis/spindle from which the feed is to be deduced is not available.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Correct part program or set the setting data 43300 ASSIGN_FEED_PER_REV_SOURCE correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

10860 Channel %1 block %2 feedrate not programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the displayed block, an interpolation type other than G00 (rapid traverse) is active. The F value has not been programmed.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Program feedrate in accordance with the interpolation type.
- G93: The feedrate is specified as a time-reciprocal value under address F in [1/min].
- G94 and G97: The feedrate is programmed under address F in [mm/min] or [m/min].
- G95: The feedrate is programmed as revolutional feedrate under address F in [mm/revolution].
- G96: The feedrate is programmed as cutting rate under address S in [m/min]. It is derived from the current spindle speed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10861 Channel %1 block %3 velocity of positioning axis %2 is zero

Parameters: %1 = Channel number
%2 = Axis
%3 = Block number, label

Definitions: No axis velocity has been programmed and the positioning velocity set in the machine data is zero.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Enter a different velocity in machine data 32060 MA_POS_AX_VELO.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10862 Channel %1 block %2 master spindle also used as path axis

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A contour has been programmed that also includes the master spindle as contouring axis. However, the velocity of the contour is derived from the rotational speed of the master spindle (e.g. G95).

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Modify the program so that no reference is possible to the program itself.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10870 Channel %1 block %2 facing axis not defined

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: When constant cutting speed is activated via the G96 function, the spindle speed is controlled through the position of the facing axis such that the cutting speed programmed under S [mm/min] is applied at the tool tip. In the channel-specific MD 20100 DIAMETER_AX_DEF[n,m]=x (n ... channel index, m ... spindle index, x ... axis name), the name of the facing axis [string] can be set for each of the 5 spindles that are used for the speed calculation.

S [rpm] = (SG96 [m/min] x 1000) : (Dfacing axis [mm] x pi)

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Enter the name of the facing axis in the channel-specific machine data 20100 DIAMETER_AX_DEF for the spindles used.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10880 Channel %1 block %2 too many empty blocks between two traversing blocks when inserting chamfers or radii

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Between 2 blocks containing contour elements and which are to be joined with a chamfer or a radius (CHF, RND), more blocks without contour information have been programmed than provided for in the machine data 20200 CHFRND_MAXNUM_DUMMY_BLOCKS.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify the part program in order that the permissible number of dummy blocks is not exceeded or adapt the channel-specific machine data 20200 CHFRND_MAXNUM_DUMMY_BLOCKS (dummy blocks with chamfers/radii) to the maximum number of dummy blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10881 Channel %1 block %2 overflow of local block buffer when inserting chamfers or radii

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Between 2 blocks containing the contour elements and to be joined with a chamfer or a radius (CHF, RND), so many dummy blocks have been programmed without contour information that the internal buffer is too small.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program such that the number of dummy blocks is reduced.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10882 Channel %1 block %2 activation of chamfers or radii (non-modal) without traversing movement in the block

Parameters: %1 = Channel number

%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: No chamfer or radius has been inserted between 2 linear or circle contours (edge breaking) because:
- There is no straight line or circle contour in the plane
- There is a movement outside of the plane
- A plane change has taken place
- The permissible number of dummy blocks without traversing information has been exceeded.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Correct the part program according to the above error description or change the number of dummy blocks in the channel-specific MD CHFRND_MAXNUM_DUMMY_BLOCKS to comply with the maximum number allowed for in the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10883 Channel %1 block %2 chamfer or fillet has to be reduced

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm is output, if at least one of the relevant blocks when inserting chamfers or radii is so short, that the contour element to be inserted must be reduced against its originally programmed value. The alarm occurs only if bit 4 is set in the machine data $MN_ENABLE_ALARM_MASK. Otherwise, the chamfer or radius is adapted without an alarm being output.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- NC Stop on alarm at block end.

Remedy: Modify NC program of continue program without modifications after CANCEL and Start or with Start alone.

Program Continuation: Clear alarm with the Delete key or NC START.

10890 Channel %1 block %2 overflow of local block buffer when calculating splines

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The maximum permissible number of empty blocks is limited by a machine data.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Modify part program
- Modify machine data
- Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10891 Channel %1 block %2 multiplicity of node is greater than its order

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: In the B spline the distance between nodes PL (node = point on spline at which 2 polynomials meet) has been programmed with zero too often in succession (i.e. the "multiplicity" of a node is too great).
In the quadratic B spline the node distance may not be specified more than twice with zero in succession, and in the cubic B spline not more than three times.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the node distance PL = 0 in succession no more than the degree of the B spline used.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10900 Channel %1 block %2 no S value programmed for constant cutting speed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If G96 is active, the constant cutting speed under address S is missing.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Program constant cutting speed under S in [m/min] or deselect the function G96. For example, with G97 the previous feed is retained but the spindle continues to rotate at the current speed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10910 Channel %1 block %2 excessive velocity of one path axis

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With active transformation, an excessive increase in velocity occurs in one or several axes, e.g. because the path passes close by the pole.

Reactions:
- Alarm display.
- Local alarm reaction.

Remedy: Divide the NC block into several blocks (e.g. 3) so that the path section with the excess is as small as possible and therefore of short duration. The other blocks are then traversed at the programmed velocity.

Program Continuation: Clear alarm with the Delete key or NC START.

10911 Channel %1 block %2 transformation prohibits to traverse the pole

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The given curve passes through the pole of the transformation.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

10912 Channel %1 block %2 preprocessing and main run might not be synchronized

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The preset positioning axis run cannot be accurately calculated beforehand. The reason for this is either that the axes involved in the transformation are traversed as positioning axes or that a transformation pole is circumnavigated too frequently by the curve. The velocity check is performed starting from this block in the main run. It is more conservative than with anticipated calculation. The LookAhead function is deactivated. If it is not possible to take over the velocity check into the main run, part program processing is aborted.

Reactions: - Alarm display.

Remedy: No action is usually necessary. The velocity control operates more effectively, however, if the part program is modified.

- If a transformation pole is circumnavigated several times by the curve, it helps to split up the block into smaller parts.
- If a positioning axis is the cause, you should check whether the axis can be traversed as a path axis. The Look Ahead function remains deactivated until preprocessing can be based on defined conditions again (e.g. as a result of change from JOG->AUTO, tool or tool edge change).

Program Continuation: Clear alarm with the Delete key or NC START.

10913 Channel %1 block %2 negative feed profile is ignored

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The given feed profile is in part negative. However, negative path feed is not allowed. The feed profile is ignored. The specified feed block end value is taken when traversing over the entire block.

Reactions: - Local alarm reaction.
- Alarm display.

Remedy: No action is usually necessary. The alarm message indicates an error in the programming, however, and this should be corrected.

Program Continuation: Clear alarm with the Delete key or NC START.

10914 Movement not possible while transformation active - in channel %1, block %2

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The machine kinematics does not allow the specified motion. Transformation-dependent error causes can be in: TRANSMIT: A (circular) area exists around the pole, where positioning is not possible. The area is caused by the fact that the tool reference point cannot be traversed as far as into the pole.

The area is defined by:

- the machine data ($MC_TRANSMIT_BASE_TOOL..)
- the active tool length compensation (see $TC_DP..). Whether the tool length compensation is included in the calculation depends on the working plane selected (see G17,..).
- The machine stops before the faulty block.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.

Remedy: Modify part program. Change the incorrectly specified tool length compensation.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

10930 Channel %1 block %2 interpolation type not allowed in stock removal contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The following types of interpolation are allowed in the contour program for stock removal: G00, G01, G02, G03, CIP, CT

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: In the contour subroutine, program only path elements that consist of straight lines and arcs.

Program Continuation: Clear alarm with the RESET key. Restart part program

10931 Channel %1 block %2 incorrect stock removal contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The following errors occurred in the subroutine for the contour during stock removal:
- Full circle
- Overlapping contour elements
- Wrong start position

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: The errors listed above must be corrected in the subroutine for the stock removal contour.

Program Continuation: Clear alarm with the RESET key. Restart part program

10932 Channel %1 block %2 preparation of contour has been restarted

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The first contour preparation/contour decoding run must be terminated with EXECUTE.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.

Remedy: Program the keyword EXECUTE to terminate the contour preparation in the part program before again calling up contour segmentation (keyword CONTPRON).

Program Continuation: Clear alarm with the RESET key. Restart part program

10933 Channel %1 block %2 contour program does not contain enough contour blocks

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The contour program contains:
- Less than 3 contour blocks with CONTPRON
- No contour blocks with CONTDCON

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Increase the size of the subroutine with the stock removal contour to include at least 3 NC blocks with movements in both axes of the current machining plane.

Program Continuation: Clear alarm with the RESET key. Restart part program

10934 Channel %1 block %2 array for contour segmentation is set too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: During contour segmentation (activated with the keyword CONTPRON), the field for the contour table has been detected as too small. For every permissible contour element (circle or straight line) there must be a row in the contour table.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Base the definition of the field variables of the contour table on the contour elements to be expected. The contour segmentation function divides up some NC blocks into as many as 3 machining cuts. Example: N100 DEF TABNAME_1 [30, 11] Field variables for the contour table provide for 30 machining cuts. The number of columns (11) is a fixed quantity.

Program Continuation: Clear alarm with the RESET key. Restart part program

10940 Channel %1 block %2 curve table %3: delete/overwrite not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: The curve table can only be deleted if it is not active in a link.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: It is necessary to deactivate all links that are being used by the curve table to be deleted.

Program Continuation: Clear alarm with the RESET key. Restart part program

10941 Channel %1 block %2 curve table %3: NC memory full

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: The free dynamic disk space is exceeded during definition of the curve table.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Delete curve tables that are no longer required, or reconfigure the disk space for the curve tables. The curve table definition process now has to be repeated; see machine data:
MD MN_MM_NUM_CURVE_TABS, MD MN_MM_NUM_CURVE_SEGMENTS, MD MN_MM_NUM_CURVE_POLYNOMS.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

10942 Channel %1 block %2 curve table %3: illegal instruction during definition

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: Various illegal command sequences cause the output of this alarm during the definition of the curve table. For example, it is impermissible to terminate definition of a curve table with M30 before programming the CTABEND command.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct the part program and start it again.

Program Continuation: Clear alarm with the RESET key. Restart part program

10943 Channel %1 block %2 curve table %3: direction reversal of lead value in the block not allowed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: The conditions for converting a programmed contour to a curve table were not fulfilled in this block.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct the part program and start it again.

Program Continuation: Clear alarm with the RESET key. Restart part program

10944 Channel %1 block %2 curve table %3: illegal transformation

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: It is impermissible to use a transformation in a curve table if the leading axis or following axis programmed in CTABDEF is involved in the transformation. Exception: TRAANG.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct NC part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

10945 Channel %1 block %2 curve table %3: illegal coupling of axes

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: It is not possible to program axis links for the leading axes and following axis programmed in CTABDEF.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Correct NC part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

10946 Channel %1 block %2 curve table %3: no contour defined

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table
Definitions: No movement for the leading axis was programmed between CTABDEF and CTABEND. A curve table is not permitted without a contour.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Correct the part program and start it again.
Program Continuation: Clear alarm with the RESET key. Restart part program

10947 Channel %1 block %2 curve table %3: contour not continuous

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table
Definitions: The contour in a curve table must be continuous. Incontinuity can occur, for example, as a result of activating a transformation.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Correct the part program and start it again.
Program Continuation: Clear alarm with the RESET key. Restart part program

10948 Channel %1 block %2 curve table %3: position jump at end of period

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table
Definitions: A periodic curve table was defined in which the position of the following axis at the end of the table was different to the position at the start of the table.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Correct the part program and start it again.
Program Continuation: Clear alarm with the RESET key. Restart part program

10949 Channel %1 block %2 curve table %3: missing master axis motion

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table
Definitions: A slave axis motion has been programmed without a master axis motion.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct the part program and start it again.

Program Continuation: Clear alarm with the RESET key. Restart part program

10950 Channel %1 calculation of arc length function too inaccurate

Parameters: %1 = Channel number

Definitions: The calculation of the arc length function could not be performed to the required accuracy.

Reactions: - Alarm display.
- Warning display.

Remedy: The calculation of the arc length function could not be performed to the required accuracy during active polynomial interpolation. Either increase MD SPLINE_FEED_PRECISION or reserve more memory for the representation of the arc length polynomials. MD MM_ARCLENGTH_SEGMENTS defines how many polynomial segments can be used per block in order to approximate the arc length function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10951 Channel %1 block %2 curve table %3: following value period is zero

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: -

Reactions: - Alarm display.

Remedy: Ensure that the table specification is correct.

Program Continuation: Clear alarm with the Delete key or NC START.

10955 Channel %1 block %2 curve table %3: missing master axis motion

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: A slave axis motion has been programmed without a master axis motion. This can also occur if, with active radius compensation, a block is created in which the slave axis moves but not the master axis. The alarm is for information only and can be suppressed by setting MD $MC_CTAB_ENABLE_NO_LEADMOTION = 2.

Reactions: - Alarm display.

Remedy: Alarm can be switched off via MD $MC_CTAB_ENABLE_NO_LEADMOTION = 2.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10956 Channel %1 block %2 curve table %3: NC memory limit DRAM reached

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: Insufficient memory in the DRAM while defining the curve table.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Delete the curve tables that are no longer required in the DRAM or reconfigure the memory space for the curve tables. The curve table must then be redefined. Machine data for memory configuration of the curve tables in DRAM:
MN_MM_NUM_CURVE_TABS_DRAM, MN_MM_NUM_CURVE_SEGMENTS_DRAM, MN_MM_NUM_CURVE_POLYNOMS_DRAM.

Program Continuation: Clear alarm with the RESET key. Restart part program

10960 Channel %1 block %2 COMPCURV/COMPCAD and radius compensation cannot be used simultaneously

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Compressor types COMPCURV and COMPCAD cannot be used in combination with tool radius compensation. Only compressor type COMPON can be activated while tool radius compensation is active.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10961 Channel %1 block %2 maximum cubic polynomials are allowed on active radius compensation.

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With active radius compensation, only up to cubic polynomials are permissible for the geometry axes. In this case no 4th or 5th degree polynomials can be programmed.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10962 Channel %1 block %2 function %3 not possible with path correction

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Function name

Definitions: With this software release, the specified function can not yet be used together with tool radius compensation. Please modify the part program or obtain a higher software version.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12000 Channel %1 block %2 address %3 programmed repeatedly

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string of the address

Definitions: Most addresses (address types) may only be programmed once in an NC block, so that the block information remains unambiguous (e.g. X... T... F... etc. - exception: G and M functions).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

- Remove from the NC program addresses that occur more than once (except for those where multiple value assignments are allowed).
- Check whether the address (e.g. the axis name) is specified via a user-defined variable (this may not be easy to see if allocation of the axis name to the variable is performed in the program through computational operations only).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12010 Channel %1 block %2 address %3 address type programmed too often

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string of the address

Definitions: For each address type, it is defined internally how often it may occur in a DIN block (for instance, all axes together form one address type for which a block limit also applies).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

The program information must be split up over several blocks (but make sure that the functions are of the non-modal type!).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12020 Channel %1 block %2 illegal address modification

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Valid address types are 'IC', 'AC', 'DC', 'CIC', 'CAC', 'ACN', 'ACP', 'CACN', 'CACP'. Not each of these address modifications can be used for each address type. The Programming Guide specifies which of these can be used for the various address types. If this address modification is applied to address types that are not allowed, then the alarm is generated, e.g.:

N10 G02 X50 Y60 I=DC(20) J30 F100

interpolation parameters with DC.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Apply non-modal address modifications only for permissible addresses, in accordance with the Programming Guide.

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12030 Channel %1 block %2 invalid parameter or data type in %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: In polynomial interpolation, polynomials must not be greater than the 3rd degree (refer to Programming Guide).
f(p) = a0 + a1 p + a2 p2 + a3 p3
The coefficients a0 (the starting points) are identical to the end points of the preceding block and need not be programmed. In the polynomial block, a maximum of 3 coefficients per axis is therefore allowed (a1, a2, a3).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12040 Channel %1 block %2 expression %3 is not of data type 'AXIS'

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string in the block

Definitions: Some keywords require that the data in their parameters be written in variables of the type "AXIS". For example, in the keyword PO the axis identifier must be specified in the parenthesized expression, and it must be defined as a variable of the AXIS type. With the following keywords only parameters of the AXIS type are possible:
AX[..], FA[..], FD[..], FL[..], IP[..], OVRA[..], PO[..], POS[..], POSA[..]
Example:
N5 DEF INT ZUSTELL=Z1 incorrect, this does not specify an axis identifier but the number 26 161
N5 DEF AXIS ZUSTELL=Z1 correct
:
N10 POLY PO[X]=(0.1,0.2,0.3) PO[Y]=(22,33,44) &PO[INFEED]=(1,2,3)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct the part program in accordance with the instructions given in the Programming Guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12050 Channel %1 block %2 DIN address %3 not configured

Parameters: %1 = Channel number
%2 = Block number, label
%3 = DIN address in the source text block

© Siemens AG, 2002. All rights reserved

Definitions: The name of the DIN address (e.g. X, U, X1) is not defined in the control. In addition to the fixed DIN addresses, the control also has variable addresses. Refer to "Variable addresses" in the Programming Guide. The names of these addresses can be altered by machine data.

e.g.: DIN identifier -> Configured identifier

G01 -> LINE, G04 -> WAIT ...

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Study the Programming Guide and the machine data with respect to the addresses actually configured and their significance and correct the DIN block accordingly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12060 Channel %1 block %2 same G group programmed repeatedly

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The G functions that can be used in the part program are divided into groups that are syntax defining or non-syntax defining. Only one G function may be programmed from each G group. The functions within a group are mutually preclusive.

The alarm refers only to the non-syntax defining G functions. If several G functions from these groups are called in one NC block, the last of these in a group is active in each case (the previous ones are ignored).

Syntax defining G functions: 1. to 4th G group

Non-syntax defining G functions: 5. to nth G group

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

No remedy is required. You should, however, check whether the G function last programmed really is the one required.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12070 Channel %1 block %2 too many syntax-defining G functions

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Syntax defining G functions determine the structure of the part program block and the addresses contained in it. Only one syntax defining G function may be programmed in each NC block. The G functions in the 1st to 4th G group are syntax defining.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Analyze NC block and distribute the G functions over several NC blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12080 Channel %1 block %2 syntax error in text %3

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source text area

© Siemens AG, 2002. All rights reserved

Definitions: At the text position shown, the grammar in the block is incorrect. The precise reason for this error cannot be specified in more detail because there are too many possibilities.

Example 1:

N10 IF GOTOF ... ; the condition for the jump is missing!

Example 2:

N10 DEF INT VARI=5

N11 X VARI ; the operation is missing for the X and VARI variables

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Analyze the block and correct it in accordance with the syntax rules given in the Programming Guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12090 Channel %1 block %2 unexpected parameter %3

Parameters:
%1 = Channel number

%2 = Block number, label

%3 = Disallowed parameters in the text

Definitions: The programmed function has been predefined; no parameters are allowed in its call. The first unexpected parameter is displayed.

Example: On calling the predefined subroutine TRAFOF (switching off a transformation) parameters have been transferred (one or more).

Reactions:
- Correction block
- Interface signals are set.
- Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Program function without parameter transfer.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12100 Channel %1 block %2 number of passes %3 not permissible

Parameters:
%1 = Channel number

%2 = Block number, label

%3 = Number of passes

Definitions: The subroutines called with MCALL are modal, i.e. after each block with positional information a routine run is automatically performed once. For this reason, programming of the number of passes under address P is not allowed.

The modal call is effective until another MCALL is programmed, either with a new subroutine name or without (delete function).

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Program the subroutine call MCALL without number of passes.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12110 Channel %1 block %2 block syntax cannot be interpreted

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The addresses programmed in the block are not permissible together with the valid syntax defining G function, e.g. G1 I10 X20 Y30 F1000.
An interpolation parameter must not be programmed in the linear block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Check the block structure and correct in accordance with the programming requirements.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12120 Channel %1 block %2 G function not separately programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The G function programmed in this block must be alone in the block. No general addresses or synchronous actions may occur in the same block. These G functions are:
G25, G26: Working area and spindle speed limitation
G110, G111, G112: Pole programming with polar coordinates
G92: Spindle speed limitation with v constant
STARTFIFO, STOPFIFO: Control of preprocessing buffer
E.g. G4 F1000 M100: no M function allowed in the G4 block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Program G function by itself in the block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12130 Channel %1 block %2 illegal tool orientation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool orientation may only be contained in a modal motion block or in a WAB block (repositioning).
It can be programmed via Euler angles (A1, B1, C1), normal vector components (A2, B2, C2), direction vectors (A3, B3, C3) or the axis end values. If the tool orientation is programmed in conjunction with the functions:
G04 (dwell time), G33 (thread cutting with constant lead), G74 (approach reference points) or REPOSL, REPOSQ, REPOSH (repositioning)
then an alarm is issued with Euler angles, direction vectors and normal vector components.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Program tool orientation with the axis end values or use a separate block for this.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12140 Channel %1 block %2 functionality %3 not implemented

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Software construct in the source text

Definitions: In the full configuration of the control functions are possible that are not yet implemented in the current version.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
The displayed function must be removed from the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12150 Channel %1 block %2 operation %3 not compatible with data type

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String (violating operator)

Definitions: The data types are not compatible with the required operation (within an arithmetic expression or in a value assignment).
Example 1:
Arithmetic operation
N10 DEF INT OTTO
N11 DEF STRING[17] ANNA
N12 DEF INT MAX
:
N50 MAX = OTTO + ANNA
Example 2:
Value assignment
N10 DEF AXIS DRILL N11 DEF INT OTTO: N50 OTTO = DRILL

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Alter the definition of the variables used such that the required operations can be executed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12160 Channel %1 block %2 range of values exceeded

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed constant or the variable exceeds the value range that has previously been established by the definition of data type.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct value of the constant or adapt data type. If the value for an integer constant is too great, it can be specified as real constant by adding a decimal point.
Example:
R1 = 9 876 543 210 Correct: R1 = 9 876 543 210.
Value range INTEGER: Value range REAL: 2-1022 to 2+1023

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12170 Channel %1 block %2 identifier %3 defined repeatedly

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Symbol in block

Definitions: The symbol shown in the error message has already been defined in the active part program. Note that user-defined identifiers may occur more than once if the multiple definition occurs in other (sub)programs, i.e. local variables may be redefined with the same name if the program has been exited (subprograms) or has already been concluded. This applies both to user-defined symbols (labels, variables) and to machine data (axes, DIN addresses and G functions).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: The symbol already known to data management is displayed. This symbol must be looked for in the definition part of the current program using the program editor. The 1st or 2nd symbol must be given a different name.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12180 Channel %1 block %2 illegal chaining of operators %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Chained operators

Definitions: Operator chaining means the writing in sequence of binary and unary operators without using any form of parentheses.
Example:
N10 ERG = VARA - (- VARB) ; correct notation
N10 ERG = VARA - - VARB ; error!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Formulate the expression correctly and unambiguously making use of parentheses. This improves clarity and readability of the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12190 Channel %1 block %2 variable of type ARRAY has too many dimensions

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Array with variables of type STRING may be no more than 1-dimensional, and with all other variables no more than 2-dimensional.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct the array definition, with multi-dimensional arrays define a second 2-dimensional array if necessary and operate it with the same field index.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12200 Channel %1 block %2 symbol %3 cannot be created

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Symbol in the source block

Definitions: The symbol to be created with the DEF instruction cannot be created because:
- it has already been defined (e.g. as variable or function)
- the internal memory location is no longer sufficient (e.g. with large arrays)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Make the following checks:
- Check with the text editor whether the name to be allocated in the active program cycle (main program and called subprograms) has already been used.
- Estimate the memory requirements for the symbols already defined and reduce these if necessary by using fewer global and more local variables.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12210 Channel %1 block %2 string %3 too long

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String in the source block

Definitions:
- In the definition of a variable of type STRING, it has been attempted to initialize more than 100 characters.
- In an allocation, it has been found that the string does not fit in the given variable.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Select shorter string or divide up the character string into 2 strings
- Define larger string variable

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12220 Channel %1 block %2 binary constant %3 in string too long

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Binary constant

Definitions: When initializing or allocating the value of a variable of type STRING more than 8 bits have been found as binary constant.
DEF STRING[8] OTTO = "ABC'H55"B000011111'DEF"

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

In the window for the alarm message, the first characters of the binary constant are always displayed although the surplus bit might not be located until further on. Therefore, the complete binary constant must always be checked for an incorrect value.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12230 Channel %1 block %2 hexadecimal constant %3 in string too long

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Hexadecimal constant

Definitions: A string can also contain bytes that do not correspond to a character that can be entered or one that is available on a keyboard with a minimized number of keys. These characters can be input as binary or hexadecimal constants. They may occupy up to 1 byte each only - therefore be < 256, e.g.

N10 DEF STRING[2] OTTO=" 'HCA' 'HFE' "

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

In the window for the alarm message, the first characters of the binary constant are always displayed although the surplus bit might not be located until further on. Therefore, the complete hexadecimal constant must always be checked for an incorrect value.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12240 Channel %1 block %2 tool orientation %3 defined repeatedly

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Text

Definitions: Only one tool orientation can be programmed per DIN block. This can either be defined via the 3 Euler angles, or the end points of the axes, or through direction vectors.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Since the tool orientation can be set in 3 different ways, the most advantageous should be selected. For this type of specification, the addresses and value assignments must be programmed and all other orientation parameters must be removed.

Axis end points (additional axes): A, B, C axis identifiers Euler angles: A2, B2, C2 Direction vectors: A3, B3, C3

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12250 Channel %1 block %2 nested macro %3 not possible

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

© Siemens AG, 2002. All rights reserved

Definitions: The macro technique supplies a 1-line instruction or series of instructions with a new identifier by means of the keyword DEFINE. No further macro may be contained in the string of instructions (nesting). Example: N10 DEFINE MACRO1 AS G01 G91 X123 MACRO2 F100

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Nested macros must be replaced by the full program information.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12260 Channel %1 block %2 too many initialization values specified %3

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: In the initialization of an array (array definition and value assignments to individual array elements) there are more initialization values than array elements. Example: N10 DEF INT OTTO[2,3]=(..., ..., {more than 6 values})

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Check the NC program to establish whether:

1. During array definition the number of array elements (n,m) was indicated correctly (DEF INT FIELDNAME[n,m] e.g. an array with 2 lines and 3 columns: n=2, m=3). 2. During initialization the value assignments have been made correctly (values of the individual field elements separated by comma, decimal point for variables of the type REAL).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12261 Channel %1 block %2 initialization of %3 not allowed

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: Frame type variables cannot be initialized in the definition. Example: DEF FRAME LOCFRAME = CTRANS(X,200)

Equally, no default values can be programmed for axes in the program run when initializing fields by SET.

Reactions:
- Correction block
- Interface signals are set.
- Alarm display.

Remedy: IPerform initialization in separate block in the execution part of the program: DEF FRAME LOCFRAME LOCFRAME = CTRANS(X,200)

When using for axis variables:

Replace DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = SET (X, , Y) by: DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = X AXIS_VAR [7] = Y

If REDEF ... INIRE, INIPO, INICF, PRLOC changes the behavior of a GUD, LUD etc., then the machine data $MN_DEFAULT_VALUES_MEM_MASK must equal 1.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12270 Channel %1 block %2 macro identifier %3 already defined

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string macro name

Definitions: The name of the macro to be selected by the instruction DEFINE is already defined in the control as:
Macro name
Keyword
Variable
Configured identifier.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Select DEFINE instruction with another macro name.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12280 Channel %1 block %2 maximum macro length %3 exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The string of instructions on the right side of the macro is limited to 256 characters. If an attempt is made to define a longer character string under one macro (possible only through V.24 input of NC blocks, because communication between operator panel and NCK is limited to a block length of 242 characters), an alarm is displayed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Divide the functions defined under the macro into 2 macros.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12290 Channel %1 block %2 arithmetic variable % 3 not defined

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string arithmetic variable

Definitions: Only the R parameters are predefined as arithmetic variables. All other arithmetic variables must be defined with the DEF instruction before being used. The number of arithmetic parameters is defined via machine data. The names must be unambiguous and may not be repeated in the control (exception: local variables).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Define the required variable in the definition part of the program (possibly in the calling program if it is to be a global variable).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12300 Channel %1 block %2 call-by-reference parameter missing on subroutine call %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: In the subroutine definition, a formal REF parameter (call-by-reference parameter) has been specified with no actual parameter assigned to it.
The assignment takes place in the subroutine call on the basis of the position of the variable name and not on the basis of the name!
Example:
Subroutine:
(2 call-by-value parameters X and Y, 1 call-by-reference parameter Z)
PROC XYZ (INT X, INT Y, VAR INT Z)
:
M17
ENDPROC
Main program:
N10 DEF INT X
N11 DEF INT Y
N11 DEF INT Z
:
N50 XYZ (X, Y) ; REF parameter Z missing
or
N50 XYZ (X, Z) ; REF parameter Z missing!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Assign a variable to all REF parameters (call-by-reference parameters) of the subroutine when calling. No variable must be assigned to "normal" formal parameters (call-by-value parameters), as these are defaulted with 0.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12310 Channel %1 block %2 axis parameter missing on procedure call %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: When calling the subroutine, an AXIS parameter is missing which, according to the EXTERN declaration, should be present.
With the EXTERN instruction, user-defined subroutines (procedures) are made "known" that have a parameter transfer.
Procedures without parameter transfer require no EXTERN declaration.
Example:
Subroutine XYZ (with the formal parameters):
PROC XYZ (INT X, VAR INT Y, AXIS A, AXIS B)
EXTERN instruction (with variable types):
EXTERN XYZ (INT, VAR INT, AXIS, AXIS) Subroutine call (with actual parameters):
N10 XYZ (, Y1, R_TABLE)
Variable X is defaulted with value 0

© Siemens AG, 2002. All rights reserved

Variable Y is supplied with the value of the variable Y1 and returns the results to the calling program after the subroutine run

Variable A is supplied with the axis in R_TABLE

Variable B missing!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Program the missing AXIS parameter in the call.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12320 Channel %1 block %2 parameter %3 is no variable

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: A constant or the result of a mathematical expression has been assigned to a REF parameter instead of a variable at the time of the subroutine call, even though only variable identifiers are allowed.

Examples:

N10 XYZ (NAME_1, 10, OTTO) or

N10 XYZ (NAME_1, 5 + ANNA, OTTO)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Remove the constant or the mathematical expression from the NC block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12330 Channel %1 block %2 type of parameter %3 incorrect

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: When calling a procedure (a subroutine) it is found that the type of the actual parameter cannot be converted into the type of the formal parameter. There are two possible cases:

- Call-by-reference parameter: Actual parameter and formal parameter must be of precisely the same type, e.g. STRING, STRING.
- Call-by-value parameter: Actual parameter and formal parameter can in principle be different providing conversion is basically possible. In the present case, however, the types are generally not compatible, e.g. STRING -> REAL.

Overview of type conversions:

- from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: -
- from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: -
- from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: -
- from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: -

© Siemens AG, 2002. All rights reserved

- from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: -
- from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: -
- from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes

1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.

2) String length 0 => FALSE, otherwise TRUE.

*) At type conversion from REAL to INT fractional values that are >=0.5 are rounded up, others are rounded down.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check transfer parameters of the subroutine call and define the application accordingly as call-by-value or call-by-reference parameter.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12340 Channel %1 block %2 number of parameters too high %3

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: When calling a function or a procedure (predefined or user-defined) more parameters were transferred than defined.

Predefined functions and procedures: The number of parameters has been set permanently in the NCK.

User-defined functions and procedures: The number of parameters is established by type and name in the definition.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check whether the correct procedure/function has been called. Program the number of parameters in accordance with the procedure/function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12350 Channel %1 block %2 parameter %3 no longer possible

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: An attempt has been made to transfer actual parameters although axis parameters located before them have not been assigned. For procedure or function calls, assignment of parameters that are no longer required can be omitted, if subsequently no further parameters are to be transferred. Example: N10 FGROUP(X, Y, Z, A, B) ; max. 8 axes possible The following call-by-value parameters would then be initialized with zero because the space-dependent assignment has been lost on account of the omitted axis parameters.

Axes that can be omitted and following parameters do not occur in the predefined procedures and functions.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. In predefined procedures and functions either remove the following parameters or transfer any preceding axis parameters. In user-defined procedures and functions, parameter transfer must be programmed in accordance with the instructions given in the machine manufacturer's programming guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12360 Channel %1 block %2 dimension of parameter %3 incorrect

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The following possibilities of error must be checked:
- The current parameter is an array, but the formal parameter is a variable
- The current parameter is a variable, but the formal parameter is an array
- The current and formal parameters are arrays, but not with the dimensions to be defined.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the NC part program in accordance with the cause of error as listed above.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12370 Channel %1 block %2 range of values %3 not permissible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: A variable has been initialized with a value range outside an initialization block. The definition of program-global variables is allowed only in special initialization blocks. These variables can be initialized with a value range.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Remove the value range specification (begins with the keyword OF) or define the variable as a global variable in the initialization block and initialize it with a value range.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12380 Channel %1 block %2 maximum memory capacity reached

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The data definitions in this block cannot be processed because the maximum available memory for creating the data has been filled, or because the data block cannot accommodate any further data.

The alarm can also occur if several subroutine calls are executed in sequence and no block with an effect on the machine is generated (motion, dwell, M function).

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Please inform the authorized personnel/service department. Reduce the number of variables, reduce the size of arrays, or increase the capacity of the data management system.

- If new macro definitions are to be introduced -> increase machine data 18160 MM_NUM_USER_MACROS
- If new GUD definitions are to be introduced -> check machine data 18150 MM_GUD_VALUES_MEM, 18130 MM_NUM_GUD_NAMES_CHAN, 18120 MM_NUM_GUD_NAMES_NCK
- If the error occurs while executing an NC part program with LUD definitions or when using cycle programs (the parameters count as LUD variable of the cycle program), the following machine data must be checked:

28040 MM_LUD_VALUES_MEM,
18242 MM_MAX_SIZE_OF_LUD_VALUE,
18260 MM_LUD_HASH_TABLE_SIZE,
28020 MM_NUM_LUD_NAMES_TOTAL,
28010 MM_NUM_REORG_LUD_MODULES

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12390 Channel %1 block %2 initialization value %3 cannot be converted

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: During initialization, a value has been assigned to a variable that does not correspond to the type of the variable, nor can it be converted to the data type of the variable.
Overview of type conversions:

- from REAL to REAL: no, INT: yes1), BOOL: yes, CHAR: yes2), STRING: -
- from INT to REAL: yes, INT: no, BOOL: yes, CHAR: yes2), STRING: -
- from BOOL to REAL: yes, INT: yes, BOOL: no, CHAR: yes, STRING: -
- from CHAR to REAL: yes, INT: yes, BOOL: yes, CHAR: no, STRING: yes
- from STRING to REAL: -, INT: -, BOOL: yes, CHAR: yes3), STRING: no

1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
2) String length 0 => FALSE, otherwise TRUE.
3) If only one character.
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

- Define variable type such that the initialization value can be assigned, or
- Select initialization value in accordance with the variable definition.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12400 Channel %1 block %2 field %3 element does not exist

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

© Siemens AG, 2002. All rights reserved

Definitions: The following causes are possible:
- Impermissible index list; an axis index is missing
- Array index does not match the definition of the variables
- An attempt was made to access a variable at array initialization via SET or REP; this attempt did not correspond to the standard access. Single character access, partial frame access, omitted indices not possible.

A nonexistent element was addressed on initializing this array.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Array initialization: Check the array index of the addressed element. The 1st array element is given the index [0,0], the 2nd array element [0,1] etc. The right array index (column index) is incremented first.

In the 2nd row, the 4th element is also addressed with the index [1,3] (the indices start at zero).
- Array definition: Check the size of the array. The1st number indicates the number of elements in the 1st dimension (number of rows), the 2nd number indicates the number of elements in the 2nd dimension (number of columns).

An array with 2 rows and 3 columns must be defined by specifying [2,3].

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12410 Channel %1 block %2 incorrect index type for %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: In assigning a value to an element of an array variable, the array index was specified in a way that is not allowed.
Only the following are allowed as array index (in square brackets):
- Axis identifier, provided the array variable was defined as data type FRAME.
- Integer values for all other data types.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct indices of the array element with respect to variable definition or define the array variable differently.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12420 Channel %1 block %2 identifier %3 too long

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The symbol to be defined or the specified jump target has a name which is longer than the 32 characters allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. The symbol to be created or the target of program jumps (label) must conform to the system specifications, that means the name must begin with 2 letters (but the 1st sign must not be "§") and may be up to a maximum of 32 characters.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12430 Channel %1 block %2 specified index is invalid

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In specifying an array index (in the array definition) an index was used that is outside the permissible range.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. Specify array index within the permissible range. Value range per array dimension: 1 - 32 767.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12440 Channel %1 block %2 maximum number of formal arguments exceeded

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In the definition of a procedure (a subroutine) or in an EXTERN instruction, more than 127 formal parameters have been specified.

Example: PROC ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, FORMPARA128, ...) EXTERN ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, FORMPARA128, ...)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. A check must be made to determine whether all parameters really have to be transferred. If so, the formal parameters can be reduced by using global variables or R parameters, or by grouping together parameters of the same type to form an array and transfer them in this form.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12450 Channel %1 block %2 label defined twice

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The label of this block already exists.

If the NC program is compiled off-line, the entire program is compiled block for block. During this procedure all multiple labels are recognized; this is not always the case with online compilation. (Only the actual program run is compiled here, i.e. program branches that are not passed through in this run are disregarded and could therefore contain programming errors).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer is positioned on the block where the displayed label occurs for the second time. Use the editor to search the part program where this label occurs for the first time, and change one of the names.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12460 Channel %1 block %2 maximum number of symbols exceeded with %3

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: The max. number of variable definitions (GUD, LUD), macro definitions, cycle programs and/or cycle parameters that the controller's data management system is able to handle has been exceeded. If this alarm occurs in conjunction with alarm 15175 (cycles were loaded again), not enough memory is available. This situation can be remedied by modifying the machine data. If this alarm occurs in conjunction with alarm 15180 (initial.ini download failed), then this alarm shows the name of the block causing the error. (For a list of names and their meaning -> please refer to alarm 6010)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Please inform the authorized personnel/service department. Reduce the symbols in the block (possibly by using the array technique or by using R parameters), or adapt the machine data (if you have access rights). $MC_MM_NUM_LUD_NAMES_TOTAL with error in LUD blocks (i.e. if more variable definitions were made in the active part programs than allowed by the MD). GUD data blocks can only cause errors as part of the 'initial.ini download' process. Macros and cycle program definitions are reloaded at each POWER ON/NCK-RESET. This means that these blocks can only cause errors in conjunction with this process. See also the explanations for alarm 6010. Particularly in relation to the reloading of cycle programs, users with the necessary expertise can check parameter %3 to find out.

- Whether the name of the cycle program has caused the error - in this case the value of machine data $MN_MM_NUM_MAX_FUNC_NAMES should be increased, or
- Whether the name of a cycle call parameter has caused the error - in this case the value of machine data $MN_MM_NUM_MAX_FUNC_PARAM should be increased.

If it is impossible to tell from parameter %3 what has caused the error, you should increase both machine data values successively until the error disappears.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12470 Channel %1 block %2 G function %3 is unknown

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Definitions: With indirectly programmed G functions, an invalid or non-allowed group number has been programmed. Allowed group number = 1. and 5 max. number of G groups. In the displayed block, a non-defined G function has been programmed. Only "real" G functions are checked, which begin with the address G, e.g. G555. "Named" G functions such as CSPLINE, BRISK etc. are interpreted as subroutine names.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. You should decide on the basis of the machine manufacturer's programming guide whether or not the displayed G function exists or is available, or whether a standard G function has been reconfigured (or introduced by an OEM). Remove G function from the part program or program function call in accordance with the machine manufacturer's programming guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12475 Channel %1 block %2 invalid G function number %3 programmed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = G code number

Definitions: A non-allowed G function number (parameter 3) has been programmed for a G group with indirect G code programming. Only the G function numbers indicated in the Programming Guide "Fundamentals", Section 12.3 "List of G functions/Path conditions" are allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12480 Channel %1 block %2 subroutine %3 already defined

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The name used in the PROC or EXTERN instruction has already been defined in another call description (e.g. for cycles).
Example:
EXTERN CYCLE85 (VAR TYP1, VAR TYP2, ...)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block. A program name must be selected that has not yet been used as identifier. (Theoretically, the parameter declaration of the EXTERN instruction could also be adapted to the existing subroutine in order to avoid the alarm output. However, it would have been defined identically twice).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12490 Channel %1 block %2 access permission level %3 is not valid

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The desired access authorization, programmed with the keyword REDEF, has not been set. The desired protection level is either beyond the permitted value range or the protection level change is not allowed.
(The REDEF instruction is only executable in INITIAL_INI blocks on SINUMERIK 840D, P1 (6/94)).
The protection level may be changed only if:

© Siemens AG, 2002. All rights reserved

1. The current protection level is equal to or higher than the level originally defined, and
2. The new protection level is to be below the level originally defined.

The higher numerical values represent the lower protection levels. The lower 4 levels (from 7 to 4) correspond to the keyswitch positions, and the upper 4 levels are associated with 4 passwords.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

- Use the REDEF instruction only in the INITIAL_INI block
- Using the operator panel, set the current protection level to at least the same level as that of the variable with the highest level
- Program protection level within the permissible value range
- Only program new protection levels that are lower than the old values

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12500 Channel %1 block %2 in this module %3 is not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The displayed keyword may not be used in this type of block and at this location (all files in the NCK are designated as blocks).
Block types:
Program block
Contains a main program or subroutine
Data block
Contains macro or variable definitions and possibly an M, H or E function
Initialization block
Contains only selected language elements for data initialization

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Remove the displayed language elements (keyword) with its parameters from this block and insert in the block provided for this purpose.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12510 Channel %1 block %2 too many machine data %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: In the part program, in the machine data file (..._TEA) and in the initialization file (..._INI), no more than 2 machine data may be used per block.
Example:

```
N ...
N 100 $MN_OVR_FACTOR_FEEDRATE [10] = 15,
      $MN_OVR_FACTOR_FEEDRATE [11] = 20
N ...
```

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Divide up the part program block into several blocks.
- If necessary, use the local variable for storing intermediate results.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12520 Channel %1 block %2 too many tool parameters %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: In the part program, in the tool offset file (..._TOA) and in the initialization file (..._INI), no more than 5 tool offset parameters may be used per block.
Example:

```
N ...
N 100 $TC_DP1 [5,1] = 130, $TC_DP3 [5,1] = 150.123,
      $TC_DP4 [5,1] = 223.4, $TC_DP5 [5,1] = 200.12,
      $TC_DP6 [5,1] = 55.02
N ...
```

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Divide up the part program block into several blocks.
- If necessary, use the local variable for storing intermediate results.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12530 Channel %1 block %2 invalid index for %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: In macro definitions, an attempt was made to define a G function with more than 3 decades or an M function with more than 2 decades as identifier of the macro.
Example:

```
_N_UMAC_DEF  DEFINE G4444 AS G01 G91 G1234
             DEFINE M333 AS M03 M50 M99
             :
             M17
```

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Modify the macro definition in accordance with the Programming Guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12540 Channel %1 block %2 Block is too long or too complex

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The maximum internal block length after translator processing must not exceed 256 characters. After editing, for example, several macros in the block or a multiple nesting, this limit can be exceeded.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Divide up the program block into several subblocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12550 Channel %1 block %2 name %3 not defined or option not installed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: The identifier displayed has not been defined before being used.
Macro: The keyword, to be defined with the DEFINE ... AS ... statement, is missing in one file:
_N_SMAC_DEF
_N_MMAC_DEF
_N_UMAC_DEF
_N_SGUD_DEF
_N_MGUD_DEF
_N_UGUD_DEF
Variable: DEF statement is missing
Program: PROC declaration is missing

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Correct the name used (writing error)
- Check definitions of variables, subroutines and macros
- Declare subroutine with EXTERN, load subroutine to SPF-Dir
- Check interface definition of subroutine
- Check options

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12552 Channel %1 block %2 tool/magazine OEM parameter not defined. Option not set. Option not set.

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed $TC_... Cx system parameter is not known in the control.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy:
- Correct the name used (writing error)
- $TC_DPCx, $TC_TPCx, $TC_MOPCx, $TC_MAPCx, $TC_MPPCx, $TC_DPCSx, $TC_TPCSx, $TC_MOPCSx, $TC_MAPCSx, $TC_MPPCSx; with x=1,...10
- These are the OEM parameters of the tools magazines, The corresponding machine data value is set to < 10, or the option 'TM OEM parameters' has not been set.
- Use correct parameter number, or - if the name cannot be changed - set machine data correction (see $MN_MM_NUM_CC_TOA_PARAM, ... $MN_MM_NUM_CCS_TOA_PARAM, ...).)
- Check the option (machine data are only effective when the option is enabled).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12560 Channel %1 block %2 programmed value %3 exceeds allowed limits

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: In a value assignment, the permissible value range of the data type has been exceeded.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Assign value within the value range of the various data types, or if necessary use another type in order to increase the size of the value range, e.g. INT ->REAL.
Value ranges of the various variable types:
- REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023)
- INT: Property: Integers with signs, value range: +/-(231-1)
- BOOL: Property: Truth value TRUE, FALSE, value range: 0,1
- CHAR: Property: 1 ASCII character, value range: 0-255
- STRING: Property: Character string (max. 100 values), value range: 0-255
- AXIS: Property: Axis addresses, value range: Axis names only
- FRAME: Property: Geometric information, value range: As for axis paths

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12570 Channel %1 block %2 too many motion synchronous actions in %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: No more than 16 actions are allowed in a block with motion synchronous action.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Reduce the number of programmed actions.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12571 Channel %1 block %2 %3 not permissible for motion synchronous action

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: The predefined subprogram %3 specified here is not allowed in a block with motion synchronous action. It may only be contained in a "normal" block.

© Siemens AG, 2002. All rights reserved

Reactions: - Correction block
- Interface signals are set.
- Alarm display.

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12572 Channel %1 block %2 %3 only permissible for motion synchronous action

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: The predefined subprogram %3 specified here is only allowed in a block with motion synchronous action. It must not be contained alone in a "normal" block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12580 Channel %1 block %2 %3 not permissible for assignment in motion synchronous action

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: The variable displayed must not be written in a motion synchronous action. Only selected variables are permitted here, e.g. DO $AA_IW[X]=10 is not allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Please inform the authorized personnel/service department.
Modify part program.
In a motion synchronous action, only certain variables are allowed.
E.g. $AA_IM, $AC_DTGPB

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12581 Channel %1 block %2 invalid read access to %3 while in motion synchronous action

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: In a motion synchronous action, the displayed variable must not be entered as a variable that is to be read on-line, i.e.
1. The displayed variable must not be written to the left of the comparison in a motion synchr. action. Only selected variables are permissible, e.g. WHEN $AA_OVR == 100 DO ....
2. In a motion synchronous action, the displayed variable must not be used as a $$ variable, e.g. WHEN $AA_IM[X] >= $$P_AD[1] DO ... DO $AC_VC = $$P_F
3. The displayed variable must not be programmed as an online evaluated parameter of a synchronous procedure, e.g. DO SYNFCT(1, $AC_PARAM[0], $SA_OSCILL_REVERSE_POS2[Z])

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Modify program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12582 Channel %1 block %2 field index %3 incorrect

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol
Definitions: $A or $V variables are assessed in real-time in motion synchronous actions, i.e. in the interpolation cycle. All other variables (e.g. user-defined variables) are still computed at block preparation. It is not permissible to index the index of a variable for block preparation with a real-time variable.
Example:
DEF INT INPUT[3]
WHEN $A_IN[1] == INPUT[$A_INA[1]] DO ...
The locally defined variable INPUT must not be indexed with a real-time variable.
Program editing:
WHEN $A_IN[1] == $AC_MARKER[$A_INA[1]] DO ...
Reactions: - Alarm display.
- Interface signals are set.
- Correction block
Remedy: Modify program: Use real-time variables.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12583 Channel %1 block %2 variable %3 no system variable

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol
Definitions: In motion synchronous actions, only special system variables are allowed on the left side of the compare operation for the assigned variable as input and result variable of SYNFCT and as input variable for PUTFTOCF. Real-time synchronous access is allowed here. The programmed variable is not a system variable.
Example:
DEF REAL OTTO, BERTA[2] DO SYNFCT(2,OTTO, $MN_...) ; Local variables or machine data are not allowed as parameter for SYNFCT.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block
Remedy: Modify part program. Local variables or machine data are not allowed as parameters for SYNFCT.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12584 Channel %1 block %2 variable %3 cannot be read synchronously with motion

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol
Definitions: In motion synchronous actions on the left side of the compare operation, only special variables are allowed as input variables of SYNFCT and as input variables for PUTFTOCF. Motion synchronous access is possible here.

© Siemens AG, 2002. All rights reserved

Example:
PUTFTOCF(1, $AA_OVR, 2, 1, 2)
The variable $AA_OVR is not allowed here.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program. For the functions SYNFCT and PUTFTOCF only certain variables are allowed, for example $AC_DTGPW.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12585 Channel %1 block %2 variable %3 cannot be changed synchronously with motion

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol

Definitions: When assigning SYNFCT in motion synchronous actions and result variables, only special variables are allowed. Real-time synchronous access is allowed here.
Example:
WHEN $AA_IM[AX1]>= 100 DO $AC_TIME=1000. The variable $AC_TIME (time from beginning of block) cannot be written

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program. Only certain variables are allowed for the function SYNFCT where real-time synchronous access is possible.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12586 Channel %1 block %2 motion synchronous action: type conflict in variable %3

Parameters: %1 = Channel number
%2 = Block number
%3 = Source symbol

Definitions: Type conversion is not possible for on-line variables $A.. or $V.., which are evaluated or written in the interpolation cycle. Only variables of the same type can be used together in logic operations or assigned to one another.
Example 1:
WHENEVER $AA_IM[X] > $A_IN[1] DO ...
An on-line variable of the REAL type (actual value) cannot be compared with a variable of the BOOL type (digital input)
The operation is possible if the following change is made:
WHENEVER $AA_IM[X] > $A_INA[1] DO ...
Example 2:
WHENEVER ... DO $AC_MARKER[1]=$AA_IM[X]-$AA_MM[X]
Improvement:
WHENEVER ... DO $AC_PARAM[1]=$AA_IM[X]-$AA_MM[X]

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program: Use variables of the same type.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

12587 Channel %1 block %2 motion synchronous action: operation/function %3 not allowed

Parameters: %1 = Channel number
%2 = Block number
%3 = Operator/function

Definitions: The specified function / operator is not permissible for logic operations of real-time variables in motion synchronous actions. The following operators/functions are permissible:
- == >= <= > < <> + - * /
- DIV MOD
- AND OR XOR NOT
- B_AND B_OR B_XOR B_NOT
- SIN COS TAN ATAN2 SQRT POT TRUNC ROUND ABS EXP LNX SPI

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12588 Channel %1 block %2 motion synchronous action: address %3 not allowed

Parameters: %1 = Channel number
%2 = Block number
%3 = Address

Definitions:
- The specified address cannot be programmed in motion synchronous action. Example: ID = 1 WHENEVER $A_IN[1]==1 DO D3
- The cutting edge from motion synchronous actions cannot be changed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

12589 Channel %1 block %2 motion synchronous action: variable %3 not allowed with modal ID

Parameters: %1 = Channel number
%2 = Block number
%3 = Variable name

Definitions: The modal ID in motion synchronous action must not be formed by means of an on-line variable.
Examples:
ID=$AC_MARKER[1] WHEN $a_in[1] == 1 DO $AC_MARKER[1] = $AC_MARKER[1]+1
This can be corrected in the following way:
R10 = $AC_MARKER[1]
ID=R10 WHEN $a_in[1] == 1 DO $AC_MARKER[1] = $AC_MARKER[1]+1
The ID in a synchronous action is always permanent, and cannot be changed in the interpolation cycle.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program: Replace the on-line variable by an arithmetic variable.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

12590 Channel %1 block %2 global user data cannot be created

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The number of global user data blocks is defined in machine data 18118 MM_NUM_GUD_MODULES.
Here, _N_SGUD_DEF corresponds to block 1, _N_MGUD_DEF corresponds to block 2, _N_UGUD_DEF corresponds to block 3, _N_GUD4_DEF corresponds to block 4 etc.
In the directory _N_DEF_DIR there is a file with definitions for global user data the block number of which is greater than the number of blocks given in the MD.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Please inform the authorized personnel/service department. Increase machine data 18118 MM_NUM_GUD_MODULES.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12600 Channel %1 block %2 invalid line checksum

Parameters: %1 = Channel number
%2 = Block number

Definitions: On processing an INI file or when executing a TEA file, an invalid line checksum has been detected.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct INI file or correct MD and create new INI file (via "upload").

Program Continuation: Switch control OFF - ON.

12610 Channel %1 block %2 accessing single character with call-by-reference parameter not possible %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: An attempt has been made to use a single character access for a call-by-reference parameter.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Temporarily store single characters in user-defined CHAR variable and transfer this.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12620 Channel %1 block %2 accessing this variable as single character not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string

Definitions: The variable is not a user-defined variable. The single character access is only allowed for user-defined variables (LUD/GUD).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Temporarily store variable in user-defined STRING, process this and put back into storage.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12630 Channel %1 block %2 skip ID/label in control structure not allowed

Parameters: %1 = Channel number
%2 = Block number
Definitions: Blocks with control structures (FOR, ENDIF, etc.) cannot be concealed and must not contain any labels.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block
Remedy: Modify part program: Reproduce skip ID via an IF query. Write the label alone in the block before the control structure block.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12640 Channel %1 block %2 invalid nesting of control structures

Parameters: %1 = Channel number
%2 = Block number
Definitions: Error in program run: Opened control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) are not terminated or there is no beginning of loop for the programmed end of loop.
Example:
LOOP ENDIF ENDLOOP
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Correct part program in such a way that all opened control structures are also terminated.
Program Continuation: Clear alarm with the RESET key. Restart part program

12641 Channel %1 block %2 maximum nesting depth of control structures exceeded

Parameters: %1 = Channel number
%2 = Block number
Definitions: Max. nesting depth control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) exceeded. At the present time, the max. nesting depth is 8.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Correct part program. If necessary, move parts to a subroutine.
Program Continuation: Clear alarm with the RESET key. Restart part program

12650 Channel %1 block %2 axis identifier %3 different in channel %4

Parameters: %1 = Channel number
%2 = Block number
%3 = Source symbol
%4 = Channel number with different axis definition
Definitions: In cycles that are preprocessed at Power On, only those geometry and channel axis identifiers may be used that exist in all channels with the same meaning. In different channels, different axis indices are assigned to the axis identifier.

© Siemens AG, 2002. All rights reserved

The axis identifiers are defined via machine data 20060 AXCONF_GEOAX_NAME_TAB and 20080 AXCONF_CHANAX_NAME_TAB. Example: C is the 4th channel axis in channel 1 and the 5th channel axis in channel 2.

If the axis identifier C is used in a cycle that is preprocessed at Power On, then this alarm is issued.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.

1. Modify machine data: Select the same identifiers for geometry and channel axes in all channels. Example: The geometry axes are called X, Y, Z in all channels. They can then also be programmed directly in preprocessed channels.

2. Do not program the axis directly in the cycle but define it as a parameter of the axis type. Example: Cycle definition:

PROC DRILL(AXIS DRILLAXIS) G1 AX[DRILLAXIS]=10 F1000 M17

Call from the main program:

DRILL(Z)

Program Continuation: Clear alarm with the RESET key. Restart part program

12660 Channel %1 block %2 motion synchronous action: variable %3 reserved for motion synchronous actions and technology cycles

Parameters: %1 = Channel number

%2 = Block number

%3 = Variable name

Definitions: The displayed variable may only be used in motion synchronous actions or in technology cycles. For example, '$R1' may only be used in motion synchronous actions. In standard part programs R parameters are programmed with R1.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

12661 Channel %1 block %2 technology cycle %3: no further subprogram call possible

Parameters: %1 = Channel number

%2 = Block number

%3 = Name of the technology cycle call

Definitions: In a technology cycle it is not possible to call a subroutine or another technology cycle.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

12700 Channel %1 block %2 contour definition programming not allowed as modal subprogram is active

Parameters: %1 = Channel number

%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: In the external language mode, a block is programmed with contour definition and a modal cycle is active at the same time. Because of unclear address assignment (e.g. R = radius for contour definition or return plane for drilling cycle) contour definition programming must not be used when a modal cycle is active.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12701 Channel %1 block %2 illegal interpolation type for contour definition active

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In one contour definition block, G01 is not active as interpolation function. In one contour definition block, the linear interpolation always has to be selected with G01. G00, G02, G03, G33 etc. are not permitted.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program. Program linear interpolation G01.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12710 Channel %1 block %2 illegal language element in external language mode

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The programmed language element is not allowed or unknown in external language mode. Only the language elements from Siemens mode which are used for subprogram calls (except for Lxx) and the language constructs for program repetition with REPEAT (UNTIL) are allowed.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Check that the language command is available in Siemens mode. Switch to Siemens mode with G290. Program the command in the next block and switch back to the external language mode in the following block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12720 Channel %1 block %2 program number for macro call (G65/G66) missing

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: During macro call with G65/G66 no program number was defined. The program number must be programmed with address "P".

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

12722 Channel %1 block %2 multiple ISO_2/3 macro or cycle calls in the block

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A mixture of cycle and macro calls are programmed in a block, e.g. cycle calls with G81 - G89 together with an M macro in the block or a G65/G66 macro call together with M macros in the block.
G05, G08, G22, G23, G27, G28, G29, G30, G50.1, G51.1, G72.1, G72.2 functions (ISO mode) also execute subroutine calls. Only one macro or cycle call can appear in an NC block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: Deactivate modal cycles or modal macro calls if one of the above mentioned G functions has been programmed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12724 Channel %1 block %2 no radius programmed for cylinder interpolation activation/ deactivation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When programming G07.1 (cylinder interpolation TRACYL), no cylinder radius has been programmed. Selection of the cylinder interpolation (TRACYL) with G07.1 C <cylinder radius> deselect with G07.1 C0. For "C" the name of the rotary axis defined in the TRACYL machine data has to be programmed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: G07.1 block, program the cylinder radius under the name of the rotary axis for the cylinder interpolation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12726 Channel %1 block %2 illegal plane selection with parallel axes

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a block with plane selection (G17 – G19), a basic axis of the coordinate system must not be programmed together with the parallel axis assigned to it.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

Remedy: For plane selection with G17, G18, G19 either program the basic axis of the coordinate system or the assigned parallel axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12728 Channel %1 block %2 distance for double turret not set

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool clearance for the double turret head in the setting data $SC_EXTERN_DOUBLE_TURRET_DIST is 0.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block

© Siemens AG, 2002. All rights reserved

Remedy: Enter tool clearance for the double turret head in the setting data $SC_EXTERN_DOUBLE_TURRET_DIST.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12730 Channel %1 block %2 no valid transformation machine data parameterized

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The machine data $MC__TRAFO_TYPE_1, $MC_TRAFO_AXES_IN_1[1], $MC_TRAFO_AXES_IN_2[1] are incorrectly set for G07.1, G12.1.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block
Remedy: Enter valid transformation identifier for TRACYL in $MC__TRAFO_TYPE_1 and the rotary axis number in $MC_TRAFO_AXES_IN_1[1] or $MC_TRAFO_AXES_IN_2[1].
Program Continuation: Clear alarm with the RESET key. Restart part program

12740 Channel %1 block %2 modal macro call %3 not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source string
Definitions: When calling a modal macro no other modal macro, modal cycle or modal subroutine may be active.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block
Remedy: Modify part program
Program Continuation: Clear alarm with the RESET key. Restart part program

14000 Channel %1 block %2 illegal end of file

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Part program was not terminated with M30, M02 or M17. This error is also signalled at the host if the environment variables NCPROG or NCUPROG have been set to a non-existing file.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: End part program with M30, M02 or M17 and start part program. Check environment variables NCPROG and NCUPROG at the host.
Program Continuation: Clear alarm with the RESET key. Restart part program

14001 Channel %1 block %2 illegal end of block

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: After system-internal data manipulation (e.g. when transferring blocks from an external source) a subfile can end without having LF as the last character.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Read out the part program, modify it with a text editor (e.g., insert blanks or comments before the displayed block), so that after reading it in again the part program has a different structure in the memory.

Program Continuation: Clear alarm with the RESET key. Restart part program

14009 Channel %1 block %2 illegal program path %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program path

Definitions: The part program command CALLPATH was called with a parameter (program path) referring to a directory which does not exist in the file system of the NCK.

Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy:
- Modify the CALLPATH instruction such that the parameter contains the complete path name of the loaded directory.
- Load the programmed directory in the file system of the NCK.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14010 Channel %1 block %2 invalid default parameter in subroutine call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a subroutine call with parameter transfer, parameters have been omitted that cannot be replaced by default parameters (call-by-reference parameters or parameters of type AXIS. The other missing parameters are defaulted with the value 0 or with the unit frame in the case of frames).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: The missing parameters must be provided with values in the subroutine call.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14011 Channel %1 block %2 program %3 not existing or will be edited

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name

Definitions: An unknown identifier (string) was found in the part program. It is therefore assumed that this is a program name. The part program indicated in a subprogram call or SETINT statement does not exist or is not released for machining.
As of SW 5, when a program is edited on the MMC, it can no longer be started with NC Start.
This alarm occurs if the PI FB-4 is being applied to a non-existent program.

Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: The alarm may have different causes:
- Typing error of the identifier stated in parameter 3
- Check subprogram call/SETINT statement or PROC statement. Reload part program and release for machining or close MMC editor.
- The PI "_N_ASUP__" FB-4, analogous to the SETINT instruction, might want to select a non-existent program
- Wrong path definition in subprogram call when subprogram is not called up via a search path but via an absolute path definition. Examples of complete path definitions: /_N_directoryName_DIR/_N_programName_SPF or /_N_WKS_DIR/_N_wpdName_WPD/_N_programName_SPF. directoryName: MPF, SPF, CUS, CMA, CST (defined directories). wpdName: user-specific identifier of workpiece directory (max. of 24 characters). programName: name of subprogram (max. of 24 characters)
- Parameter 3 can be a macro name. The macro definition file has an inappropriate content or it is not stored in the directory DEF_DIR or it has not been set active (via POWERON or via MMC operating step or by PI service 'F_COPY').
- Parameter 3 can be a GUD variable. There is no GUD definition file defining the variable or it is not stored in the directory DEF_DIR or it has not been set active (via the INITIAL_INI procedure or via MMC operating step or by PI service 'F_COPY').
- The part program could also be started from the CPU (ASUP).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14012 Channel %1 block %2 maximum subroutine level exceeded

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The maximum nesting depth of 8 program levels has been exceeded.
Subroutines can be called from the main program, and these in turn may have a nesting depth of 7.
In interrupt routines the maximum number of levels is 4!

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify the machining program so that the nesting depth is reduced, e.g. using the editor copy a subroutine of the next nesting level into the calling program and remove the call for this subroutine. This reduces the nesting depth by one program level.

Program Continuation: Clear alarm with the RESET key. Restart part program

14013 Channel %1 block %2 number of subroutine passes invalid

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a subroutine call the programmed number of passes P is zero or negative.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program number of passes between 1 and 9 999.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14014 Channel %1 selected program %3 not available or will be edited

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name

© Siemens AG, 2002. All rights reserved

Definitions: The selected part program is not in the NCK memory or the access authorization for the program selection is from a higher level than the current control status.
During creation, this program received the protection level of the NC control which was active at the time.
As of SW 5, when a program is edited on the MMC, it can no longer be started with NC Start.
Reactions: - Alarm display.
Remedy: Reload the program in the NCK memory or check and correct the name of the directory (workpiece overview) and the program (program overview) and reselect.
Program Continuation: Clear alarm with the Delete key or NC START.

14015 Channel %1 block %2 program %3 is not enabled

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name
Definitions: The user has no execution authorization for the file, the file is not released.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Change user authorization, release file.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14016 Channel %1 block %2 error when calling the subroutine via M/T function

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The following conflict was detected in a subprogram call per M or T function:
In the block referenced by parameter %2:
- An M or T function replacement has already been activated
- A modal subprogram call is active
- A subprogram return jump is programmed
- An end of program is programmed
- An M98 subprogram call is active (only in external language mode)
- T function replacement by D function programming in the same part program line is not possible with active TLC (G43/G44) in ISO2 system.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: An M or T function replacement is only possible if a subprogram call or return jump has not already been performed as a result of other program constructs. The part program must be corrected accordingly.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14017 Channel %1 block %2 syntax error when calling the subroutine via M function

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: When calling M code subroutine with parameter transfer, an illegal syntax was detected:
- Address extension not programmed as a constant.
- M function value not programmed as a constant.

© Siemens AG, 2002. All rights reserved

Note: If a parameter transfer has been programmed via MD $MN_M_NO_FCT_CYCLE_PAR for an M function replacement, the following restriction applies to this M function: both the address extension and the M function value must be programmed for replacement as constants.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Change the programming of the M function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14020 Channel %1 block %2 wrong value or wrong number of parameters on function or procedure call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:
- An illegal parameter value was specified in a function or procedure call.
- An illegal number of actual parameters was programmed in a function or procedure call.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14021 Channel %1 block %2 wrong value or wrong number of parameters on function or procedure call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:
- An illegal parameter value was specified in a function or procedure call.
- An illegal number of actual parameters was programmed in a function or procedure call.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

14025 Channel %1 block %2 motion synchronous action: illegal modal ID

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In modal motion synchronous actions an illegal ID number has been assigned.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

14026 Channel %1 block %2 motion synchronous action: invalid polynomial number in the FCTDEF command

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: An FCTDEF command was programmed with a polynomial number that exceeds the maximum value set in $MC_MM_NUM_FCTDEF_ELEMENTS.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

14030 Channel %1 block %2 combine OSCILL and POSP during oscillation with infeed motion

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: When oscillating controlled by synchronized actions, the assignment of oscillating and infeed axis (OSCILL) as well as the definition of the infeed (POSP) must be carried out in one NC block.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14033 Channel %1 block %2 involute: no end point programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No end point was programmed for the involute. This is either possible via direct programming with the geometry axis identifiers or by specifying the angle between start and end vector.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14034 Channel %1 block %2 involute: angle of rotation too large

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With programming of the angle of rotation (with AR) for involute interpolation, the maximum programmable angle of rotation is limited if the involute is moving towards the basic circle. The maximum value is reached if the involute touches the basic circle. With MD_INVOLUTE AUTO_ANGLE_RESTRICTION = TRUE, each angle is accepted without an alarm; if necessary, the angle is automatically limited during interpolation.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14035 Channel %1 block %2 involute: start point invalid

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: With involute interpolation, the start point of the involute must be outside the basic circle. The programmed center point or radius must be adapted accordingly.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14036 Channel %1 block %2 involute: end point invalid

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With involute interpolation, the end point of the involute must be outside the basic circle. The programmed center point / radius or end point must be adapted accordingly.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14037 Channel %1 block %2 involute: radius invalid

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With involute interpolation, the programmed radius of the basic circle must be greater than zero.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14038 Channel %1 block %2 involute not definable: end point error

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed end point does not lie on the involute defined by the start point, radius and center point of the basic circle. The deviation of the effective end radius from the programmed value is greater than the permissible value specified in MD INVOLUTE_RADIUS_DELTA.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14039 Channel %1 block %2 involute: end point programmed several times

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With involute interpolation, either the end point with the geometry axis identifiers or the angle of rotation with AR=value can be programmed. Simultaneous programming of end point and angle of rotation in one block is not allowed, since the end point can thus not be defined exactly.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14040 Channel %1 block %2 error in end point of circle

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In circular interpolation, either the circle radii for the initial point and the end point are further apart, or the circle center points are further apart, than specified in the machine data.

1. In circle radius programming the starting and end points are identical, thus the circle position is not determined by starting and end points.

2. Radii: The NCK calculates from the present start point and the other programmed circle parameters the radii for the start and the end point.

An alarm message is issued if the difference between the circle radii is either

- greater than the value in the MD 21000 CIRCLE_ERROR_CONST (for small radii, if the programmed radius is smaller than the quotient of the machine data CIRCLE_ERROR_CONST divided by 21010 CIRCLE_ERROR_FACTOR), or
- greater than the programmed radius multiplied by the MD CIRCLE_ERROR_FACTOR (for large radii, if the programmed radius is greater than the quotient of the machine data CIRCLE_ERROR_CONST divided by CIRCLE_ERROR_FACTOR).

3. Center points: A new circle center is calculated using the circle radius at the starting position. It lies on the mid-perpendicular positioned on the connecting straight line from the starting point to the end point of the circle. The angle in the radian measure between both straight lines from the starting point to the center calculated/programmed as such must be lower than the root of 0.001 (corresponding to approx. 1.8 degrees).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Check MD 21000 CIRCLE_ERROR_CONST and 21010 CIRCLE_ERROR_FACTOR. If the values are within reasonable limits, the circle end point or the circle mid-point of the part program block must be programmed with greater accuracy.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14045 Channel %1 block %2 error in tangential circle programming

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The alarm may have the following causes:

The tangent direction is not defined for tangent circle, e.g. because no other travel block has been programmed before the current block. No circle can be formed from start and end point as well as tangent direction because - seen from the start point - the end point is located in the opposite direction to that indicated by the tangent.

It is not possible to form a tangent circle since the tangent is located perpendicular to the active plane.

In the special case in which the tangent circle changes to a straight line, several complete circular revolutions were programmed with TURN.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14048 Channel %1 block %2 wrong number of revolutions in circle programming

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the circle programming, a negative number of full revolutions has been specified.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

14050 Channel %1 block %2 nesting depth for arithmetic operations exceeded

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: For calculating arithmetic expressions in NC blocks, an operand stack with a fixed set size is used. With very complex expressions, this stack can overflow.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Divide up complex arithmetic expressions into several simpler arithmetic blocks.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14051 Channel %1 block %2 arithmetic error in part program

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: • In calculating an arithmetic expression, an overflow has occurred (e.g. division by zero)
• In a data type, the representable value range has been exceeded
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Analyze the program and correct the defective point in the program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14060 Channel %1 block %2 invalid skip level with differential block skip

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With "Differential block skip", a skip level greater than 7 has been specified. (In packet 1 specification of a value for the skip level is rejected by the converter as a syntax error, i.e. the only possibility is a "Suppress block" ON/OFF on one level).
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Enter a skip level (number behind the slash) less than 8.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14070 Channel %1 block %2 memory for variables not sufficient for subroutine call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A called subroutine cannot be processed (opened), either because the internal data memory to be created for general purposes is not large enough, or because the available memory for the local program variables is too small. The alarm can only occur in MDA mode.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Analyze the part program section:
1. Has the most useful data type always been selected in the variable definitions? (For example REAL for data bits is poor; BOOL would be better)
2. Can local variables be replaced by global variables?

Program Continuation: Clear alarm with the RESET key. Restart part program

14080 Channel %1 block %2 jump destination %3 not found

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Jump destination

Definitions: In conditional and unconditional jumps, the jump destination within the program must be a block with a label (symbolic name instead of block number). If no jump destination has been found with the given label when searching in the programmed direction, an alarm is output.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check NC part program for the following possible errors:
1. Check whether the target designation is identical with the label.
2. Is the jump direction correct?
3. Has the label been terminated with a colon?

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14082 Channel %1 block %2 label %3 program section not found

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Start or end label

Definitions: The start point for repetition of the program part with CALL <program name> BLOCK <start label> TO <end label> has not been found or the same program part repetition has been called recursively.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check the start and end labels for programming repetition in the user program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14085 Channel %1 block %2 instruction not allowed

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The instruction 'TML()' may only be used in the subprogram, which replaces the T command.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14088 Channel %1 block %2 axis %3 doubtful position

Parameters:
%1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: An axis position larger than 3.40e+38 increments has been programmed. This alarm can be suppressed with bit 11 in $MN_SUPPRESS_ALARM_MASK.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14090 Channel %1 block %2 illegal D number

Parameters:
%1 = Channel number
%2 = Block number, label

Definitions: A value less than zero has been programmed under address D.

A set of parameters with 25 correction values has been automatically assigned to each active tool. Each tool can have 9 sets of parameters (D1 - D9, initial setting is D1). When the D number changes, the new parameter set is active (D0 is used for deselecting the correction values).

```
N10 G.. X... Y... T15  ; Parameter set D1 of T15 active
N50 G.. X... D3 M..    ; Parameter set D3 of T15 active
N60 G.. X.. T20        ; Parameter set D1 of T20 active
```

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program D numbers in the permissible value range (D0, D1 to D9).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14091 Channel %1 block %2 illegal function, index %3 %3

Parameters:
%1 = Channel number
%2 = Block number, label
%3 = Index

Definitions: A function was programmed which is not allowed in the current program context. The code of the function in question is entered in "index":

Index == 1: "RET" command was programmed in the main program level

Index == 2: Conflict between "cancel level"/"clear number of passes" and "implicit GET"

Index == 3: ASUP conflict start immediately after selection of overstore (up to P3)

Index == 4: MD MN_G53_TOOLCORR = 1 : SUPA/G153/G53 programmed in G75

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Index == 1: Substitute "RET" command with M17/M30

Index == 2: Insert an auxiliary block (e.g. M99) after the subroutine call to which the "cancel level"/"clear number of passes" refers

Index == 3: Overstore an auxiliary block (e.g. M99), then start ASUP (up to P3)

Index == 4: With MD MN_G53_TOOLCORR = 1: Do not activate SUPA/G53/G153 in the G75 block

Program Continuation: Clear alarm with the RESET key. Restart part program

14092 Channel %1 block %2 axis %3 is wrong axis type

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Axis name, spindle number

Definitions: One of the following three programming errors has occurred:

1. The keyword WAITP(x) "Wait with block change until the specified positioning axis has reached its end point" has been used for an axis that is not a positioning axis.

2. G74 "Reference point approach from the program" has been programmed for a spindle. (Only axis addresses are permitted).

3. The keyword POS/POSA has been used for a spindle. (The keywords SPOS and SPOSA must be programmed for the spindle positions).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the part program depending on which of the above errors is involved.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14093 Channel %1 block %2 path interval <= 0 with polynominal interpolation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In the polynomial interpolation POLY, a negative value or zero has been programmed under the keyword for the polynomial length PL=...

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

Correct the value given in PL = ...

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14094 Channel %1 block %2 polynominal degree greater than 3 programmed for polynominal interpolation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The polynomial degree in the polynomial interpolation is based on the number of programmed coefficients for an axis. The maximum possible polynomial degree is 3, i.e. the axes are according to the function:

f(p) = a0 + a1 p + a2 p2 + a3 p3

© Siemens AG, 2002. All rights reserved

The coefficient a0 is the actual position at the start of interpolation and is not programmed!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Reduce the number of coefficients. The polynomial block may have a form no greater than the following:

N1 POLY PO[X]=(1.11, 2.22, 3.33) PO[Y]=(1.11, 2.22, 3.33)

N1 PO[n]=... PL=44

n ... n ... axis identifier, max. 8 path axes per block

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14095 Channel %1 block %2 radius for circle programming too small

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The radius entered for radius programming is too small, i.e. the programmed radius is smaller than half of the distance between start and end point.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14096 Channel %1 block %2 illegal type conversion

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: During the program run, a variable value assignment or an arithmetic operation has caused data to be processed in such a way that they have to be converted to another type. This would lead to the value range being exceeded.

Value ranges of the various variable types:

- REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023)
- INT: Property: Integers with signs, value range: +/-(231-1)
- BOOL: Property: Truth value TRUE, FALSE, value range: 0,1
- CHAR: Property: 1 ASCII character, value range: 0-255
- STRING: Property: Character string (max. 100 values), value range: 0-255
- AXIS: Property: Axis addresses, value range: Axis names only
- FRAME: Property: Geometric information, value range: As for axis paths

Overview of type conversions:

- from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: -
- from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: -
- from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: -
- from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: -
- from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: -
- from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: -
- from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes

© Siemens AG, 2002. All rights reserved

1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
2) String length 0 => FALSE, otherwise TRUE.
3) If only one character.
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify the program section such that the value range is not exceeded, e.g. by a modified variable definition.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14097 Channel %1 block %2 string cannot be converted to AXIS type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The called function AXNAME - conversion of the transferred parameters of the STRING type to an axis name (return value) of the AXIS type - has not found this axis identifier in the machine data.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Check the transferred parameters (axis name) of the function AXNAME to determine whether a geometry, channel or machine axis of this name has been configured by means of the machine data:
10 000: AXCONF_MACHAX_NAME_TAB
20 070: AXCONF_GEOAX_NAME_TAB
20 080: AXCONF_CHANAX_NAME_TAB
Select the transfer string in accordance with the axis name and change the axis name in the machine data if necessary. (If a change of name is to take place via the NC part program, this change must first be validated by means of a "Power On").

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14098 Channel %1 block %2 conversion error: no valid number found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The string is not a valid INT or REAL number.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program. If it is an entry, then you can check whether the string is a number via the preset function ISNUMBER (with the same parameter).

Program Continuation: Clear alarm with the RESET key. Restart part program

14099 Channel %1 block %2 result in string concatenation too long

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The result of string chaining returns a string which is greater than the maximum string length laid down by the system.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Adapt part program. With the function STRLEN, it is also possible to test the size of the sum string before performing the chaining operation.

Program Continuation: Clear alarm with the RESET key. Restart part program

14100 Channel %1 block %2 orientation transformation not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Four transformation groupings (transformation types) can be set for each channel via machine data. If the keyword TRAORI(n) (n ... number of the transformation grouping) is used to address a transformation grouping for which the machine data is not defaulted, the alarm message will be triggered.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Press the NC Stop key and select the function "Correction block" with the soft key PROGRAM CORRECT. The correction pointer positions on the incorrect block.

- Check the number of the transformation grouping when calling the part program with the keyword TRAORI(n) (n ... number of the transformation grouping).
- Enter the machine data for this transformation grouping and then activate by "Power On".

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14101 Channel %1 block %2 orientation transformation not active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Euler angles or a vector have been used in programming an orientation and no orientation transformation is active, i.e. the keyword TRAORI(n) (n ... number of transformation grouping) is missing.
Example of correct transformation programming:

```
N100 ... TRAORI(1)
N110 G01 X... Y... ORIWKS
N120 A3... B3... C3...
N130 A3... B3... C3...
 :
N200 TAFOOF
```

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Before the transformation is applied, the number of the transformation grouping must be specified with the keyword TRAORI(n) (n is between 1 and 4).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14102 Channel %1 block %2 polynominal degree greater than 5 programmed for orientation vector angle

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: During polynomial interpolation for the orientation vector, a polynomial degree larger than 5 has been programmed.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14110 Channel %1 block %2 Euler angles and orientation vector components programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An orientation has been programmed with Euler angles and the component of an orientation vector at the same time.
Example:
N50 TRAORI (1)
N55 A2=10 B2=20 C3=50 ; alarm, because Euler angle and orientation vector
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Program only one type, in other words when transformation is switched on program either Euler angles only or orientation vectors (direction vectors) only.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14111 Channel %1 block %2 Euler angles, orientation vector and transformation axes programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An orientation has been programmed at the same time as Euler angles or components of an orientation vector and the machine axis influenced by the orientation.
Example:
N50 TRAORI (1)
N55 A2=70 B2=10 C2=0 X50 ; alarm, because Euler angle and axes were programmed
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Program only one type, in other words with transformation switched on program either Euler angles only or orientation vectors (direction vectors) only or deselect transformation (TRAFOOF) and set tool orientation by programming the auxiliary axes.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14112 Channel %1 block %2 programmed orientation path not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In 5-axis transformation, the two orientation axes place a coordinate system comprising lengths and circles of latitude on a spherical surface.
If the interpolation traverses the pole point, only the 5th axis will move and the 4th axis will retain its starting position. If a motion is programmed that does not traverse the pole point directly, but passes it very closely, the preset interpolation will be deviated from if the path forms a circle that is defined by the machine data: 24530 TRAFO5_NON_POLE_LIMIT_1 (changeover angle that refers to the 5th axis).

© Siemens AG, 2002. All rights reserved

The interpolated contour is then placed through the pole (in the immediate vicinity of the pole, the 4th axis would otherwise have to accelerate most rapidly and then decelerate again).
For the 4th axis, the result is a position deviation as compared to the programmed value. The maximum permissible angle which the programmed and the interpolated path may include is stored in the MD 24540 TRAFO5_POLE_LIMIT.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: In the vicinity of the pole, always make use of axis programming. Programming of tool orientations close to the pole should generally be avoided because this always leads to problems concerning dynamic response.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14113 Channel %1 block %2 programmed lead angle too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No further explanation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14114 Channel %1 block %2 programmed tilt angle too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No further explanation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14115 Channel %1 block %2 illegal definition of workpiece surface

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The surface normal vectors programmed at the beginning of block and at the end of block point in opposite directions.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14116 Channel %1 block %2 absolute orientation programmed while ORIPATH is active

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The orientation has been entered as an absolute value (e.g. by a direction vector) although ORIPATH is active. When ORIPATH is active, the orientation is determined from the leading and sidewards angles relative to the path tangent and surface normal.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14117 Channel %1 block %2 no angle or direction of the cone programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With taper circumference interpolation of orientation (ORICONCW and ORICONCC), either the opening angle or the direction vector of the taper must be programmed. Otherwise, the change of orientation is not clearly defined.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14118 Channel %1 block %2 no end orientation programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With taper circumference interpolation of orientation, no end orientation has been programmed. The change of orientation is therefore not clearly defined.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14119 Channel %1 block %2 no intermediate orientation programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With taper circumference interpolation of orientation with ORICONIO, an intermediate orientation must also be programmed in addition to the end orientation.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14120 Channel %1 block %2 plane determination for programmed orientation not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed orientation vectors (direction vectors) in the beginning of block and end of block point include an angle of 180 degrees. Therefore the interpolation plane cannot be determined.

© Siemens AG, 2002. All rights reserved

Example:
N50 TRAORI (1)
N55 A3=0 B3=0 C3=1
N60 A3=0 B3=0 C3=-1 ; the vector of this block is precisely opposite to that in the preceding block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify the part program so that the orientation vectors of a block are not directly opposed to each other, for instance by dividing the block up into 2 subblocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14122 Channel %1 block %2 angle and direction of the cone programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With taper circumference interpolation of orientation with ORICONCW and ORICC, only the opening angle or the direction of the taper may be programmed. Programming of both in one single block is not allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14123 Channel %1 block %2 nutation angle of the cone too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With taper circumference interpolation, the programmed opening angle of the taper must be greater than the half of the angle between the start and end orientation. Otherwise, a taper cannot be defined.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14124 Channel %1 block %2 start tangent for orientation is zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: With taper circumference interpolation with tangential continuation (ORICONTO), the start tangent of orientation must not be zero.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14125 Channel %1 block %2 programmed rotation is not possible

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The programmed rotation of tool orientation cannot be traversed.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14129 Channel %1 block %2 orientation angles and orientation vector components programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An orientation angle and components of an orientation vector were programmed at the same time.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14130 Channel %1 block %2 too many initialization values given

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: On assigning an array by means of SET, more initialization values than existing array elements have been specified in the program run.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Reduce the number of initialization values.
Program Continuation: Clear alarm with the RESET key. Restart part program

14131 Channel %1 block %2 orientation axes and lead/tilt angles programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An orientation angle and a leading or sideways angle were programmed at the same time.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14132 Channel %1 block %2 orientation axes incorrectly configured

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The configuration of the orientation axes does not match the machine kinematics.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Adapt machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

14133 Channel %1 block %2 G code for orientation definition not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: It is only possible to program a G code of the 50th G code group if machine data ORI_DEF_WITH_G_CODE is set to TRUE.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Adapt machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

14134 Channel %1 block %2 G code for orientation interpolation not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: It is only possible to program a G code of the 51st G code group if machine data ORI_DEF_WITH_G_CODE or ORI_IPO_WITH_G_CODE is set to TRUE.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Adapt machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

14140 Channel %1 block %2 position programming without transformation not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Position information was programmed for an axis position but no transformation was active.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify the program.
Program Continuation: Clear alarm with the RESET key. Restart part program

14144 Channel %1 block %2 PTP movement not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The PTP G code was programmed for a movement other than G0 or G1.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify the program.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

14146 Channel %1 block %2 CP or PTP movement without transformation not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The CP or PTP G code was programmed for a movement but no transformation was active.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify the program.

Program Continuation: Clear alarm with the RESET key. Restart part program

14148 Channel %1 illegal reference system for Cartesian manual traverse

Parameters: %1 = Channel number

Definitions: In the setting data SC_CART_JOG_MODE, an illegal value has been entered for the reference system with Cartesian manual travel.

Reactions: - Alarm display.

Remedy: Enter a permitted value in the setting data SC_CART_JOG_MODE.

Program Continuation: Clear alarm with the RESET key. Restart part program

14150 Channel %1 block %2 illegal tool carrier number programmed or declared (MD)

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A toolholder number was programmed which is negative or greater than the machine data MC_MM_NUM_TOOL_CARRIER.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Program valid toolholder number or adapt machine data MC_MM_NUM_TOOL_CARRIER.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14151 Channel %1 block %2 illegal tool carrier rotation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A toolholder was activated with an angle of rotation unequal to zero, although the associated axis is not defined. A rotary axis is not defined when all three direction components are zero.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Set angle of rotation to zero, or define the associated rotary axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14152 Channel %1 block %2 tool carrier: invalid orientation. Error code: %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Error code

© Siemens AG, 2002. All rights reserved

Definitions: An attempt was made to define a tool orientation by means of the active frame which cannot be reached with the current toolholder kinematics. This case can always occur when both rotary axes of the toolholder are not perpendicular to one another or when the toolholder has fewer than two rotary axes.

The error code has the following meaning:

1: 1. Rotary axis of the first solution violates the lower limit
2: 1. Rotary axis of the first solution violates the upper limit
10: 2. Rotary axis of the first solution violates the lower limit
20: 2. Rotary axis of the first solution violates the upper limit
100: 1. Rotary axis of the second solution violates the lower limit
200: 1. Rotary axis of the second solution violates the upper limit
1000: 2. Rotary axis of the second solution violates the lower limit
2000: 2. Rotary axis of the second solution violates the upper limit
3: The required orientation cannot be set with the given axis configuration
3: Several of the error codes that indicate a violation of the axis limits can occur simultaneously.

As when an axis limit is violated an attempt is made, by addition or subtraction of multiples of 360 degrees, to reach a valid position within the permissible axis limits, if this is not possible, it is not unequivocally defined whether the lower or upper axis limit has been violated.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Change the toolholder definition or activate another frame.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14153 Channel %1 block %2 unknown tool carrier type: %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Tool carrier type

Definitions: An invalid tool carrier type was specified in $TC_CARR23[]. Only the following are allowed: t, T, p, P, m, M.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- Interpreter stop

Remedy: Change the tool carrier data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14154 Channel %1 block %2 The amount of fine correction in parameter %3 of the orientable toolholder %4 is too large

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Invalid parameter of the orientable toolholder
%4 = Number of the orientable toolholder

Definitions: The maximum permissible value of the fine correction in an orientable toolholder is limited by the machine data $MC_TOCARR_FINE_LIM_LIN for linear variables, and by the machine data $MC_TOCARR_FINE_LIM_ROT for rotary variables. The alarm can only occur if the setting data $SC_TOCARR_FINE_CORRECTION is not equal to zero.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Enter a valid fine correction value.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14155 Channel %1 block %2 invalid base frame definition for tool carrier offset

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If a tool carrier selection causes a change in the table offset, a valid base frame must be defined in order to store this offset; for more information see machine data 20184 (TOCARR_BASE_FRAME_NUMBER).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- Interpreter stop

Remedy: Change the NC program or machine data 20184 (TOCARR_BASE_FRAME_NUMBER).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14156 Channel %1 toolholder selection error at reset

Parameters: %1 = Channel number

Definitions: The settings in RESET_MODE_MASK require that an active orientable toolholder is maintained after the reset. This is done by deselecting the old orientable toolholder and then reselecting it with data that may have been modified. If an error occurs during the reselection, this alarm is issued (as a warning) and then an attempt is made to select the orientable toolholder in the initial setting. If this second attempt is successful, the reset cycle is continued without any further alarms.

Typically, the alarm only occurs when the old orientable toolholder has been selected with TCOFR, and its axis directions have been changed in such a way before the reset that a setting suitable for the associated frame is no longer possible. If there is another cause for the alarm, this results in an alarm also being issued when attempting to select in the initial setting. This is then also displayed in plain text.

Reactions: - Alarm display.

Remedy: Check the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14157 Channel %1 block %2 illegal interpolation type with MOVT

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Linear or spline interpolation must be active with MOVT (G0, G1, ASPLINE, BSPLINE, CSPLINE).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- Interpreter stop

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14159 Channel %1 block %2 more than two angles programmed with ROTS or AROTS

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Frame rotations are described using space angles with the language commands ROTS or AROTS. A maximum of two angles can be programmed.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- Interpreter stop

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14160 Channel %1 block %2 tool length selection without geometry axis specification

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If variant C (tool length acts on the programmed axis) is activated by machine data $MC_TOOL_CORR_MODE for tool length compensation with H word and G43/G44 in ISO_2 mode, at least one geometry axis must be specified.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Change machine data $MC_TOOL_CORR_MODE or the part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14165 Channel %1 block %2 active T number does not match selected tool

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When an H or D number is programmed in ISO_2 mode, the T number used in the definition for this tool is determined. This number must be identical to the T number programmed explicitly.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Change machine data $MC_TOOL_CORR_MODE or the part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14170 Channel %1 block %2 illegal interpolation type with tool length compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If tool compensation (G43/G44) is activated in language mode ISO_2, the linear type of interpolation must be active.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14180 Channel %1 block %2 H number is not defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The specified H number is not assigned to a tool (ISO_2).
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14185 Channel %1 block %2 D number is not defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The specified D number is not assigned to a tool (language mode ISO_2).
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14190 Channel %1 block %2 H number with G49

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: G49 (select tool length compensation) and an H word not equal to H0 have been programmed simultaneously.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14195 Channel %1 block %2 D number with G49

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: G49 (select tool length compensation) and an D word not equal to D0 have been programmed simultaneously.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14197 Channel %1 block %2 D number and H number programmed simultaneously

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A D word and H word have been programmed simultaneously.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14198 Channel %1 block %2 illegal change of tool direction with tool offset

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: If an offset is active in the tool direction, block change is not possible if this would change the assignment of the offset axes to the channel axes (plane change, tool change, cutter <=> turning tool, geometry axis replacement).
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: • Modify part program.
• Reduce the offset in tool direction to zero.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14199 Channel %1 block %2 illegal plane change for tool with diameter component

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: If a tool has a wear or length component which is evaluated as a diameter for the facing axis (bit 0 and/or bit 1 in MD $MC_TOOL_PARAMETER_DEF_MASK is set) and bit 2 of this MD is also set, this tool may only be used in the plane active on tool selection. A plane change results in an alarm.
Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: • Modify part program.
• Reset bit 2 in MD $MC_TOOL_PARAMETER_DEF_MASK.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14200 Channel %1 block %2 negative polar radius

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: In the endpoint specification of a traversing block with G00, G01, G02 or G03 in polar coordinates, the polar radius entered for the keyword RP=... is negative.

Definition of terms:

- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct NC part program - permissible inputs for the pole radius are only positive absolute values that specify the distance between the current pole and the block end point. (The direction is defined by the polar angle AP=...).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14210 Channel %1 block %2 polar angle too large

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In specifying the endpoints in a traversing block with G00, G01, G02 or G03 in polar coordinates, the value range of the polar angle programmed under the keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.

Definition of terms:

- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... referred to the last programmed point in the plane, G111 ... referred to the zero point of the current workpiece coordinate system (WCS), G112 ... referred to the last pole.

Reactions:
- Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14250 Channel %1 block %2 negative pole radius

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In redefining the pole with G110, G111 or G112 in polar coordinates, the pole radius specified under keyword RP=... is negative. Only positive absolute values are permitted.

Definition of terms:

- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Correct the NC part program. Permissible inputs for the pole radius are only positive, absolute values that specify the distance between the reference point and the new pole. (The direction is defined with the pole angle AP=...).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14260 Channel %1 block %2 pole angle too large

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In redefining the pole with G110, G111 or G112 in polar coordinates, the value range of the pole angle specified under keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.

Definition of terms:

- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14270 Channel %1 block %2 pole programmed incorrectly

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: When defining the pole, an axis was programmed that does not belong to the selected processing level. Programming in polar coordinates always refers to the plane activated with G17 to G19. This also applies to the definition of a new pole with G110, G111 or G112.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC part program. Only the two geometry axes may be programmed that establish the current machining plane.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14280 Channel %1 block %2 polar coordinates programmed incorrectly

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The end point of the displayed block has been programmed both in the polar coordinate system (with AP=..., RP=...) and in the Cartesian coordinate system (axis addresses X, Y,...).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC part program - the axis motion may be specified in one coordinate system only.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14290 Channel %1 block %2 polynominal degree greater than 5 programmed for polynominal interpolation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A polynominal degree greater than five was programmed for the polynominal interpolation. You can only program polynomials up to the 5th degree.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14300 Channel %1 block %2 overlaid handwheel motion activated incorrectly

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Handwheel override has been called up incorrectly:
- 1. For positioning axes:
- Handwheel override programmed for indexing axes,
- No position programmed,
- FA and FDA programmed for the same axis in the block.
- 2. For contouring axes:
- No position programmed,
- G60 not active,
- 1. 1st G group incorrect (only G01 to CIP).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14310 Handwheel %1 configuration incorrect or inactive

Parameters: %1 = Handwheel number

Definitions:
- The inputs are using a drive with a drive number that does not exist or
- an inactive drive for assignment of the handwheel (ENC_HANDWHEEL_MODULE_NR) or
- an axis is using a measuring circuit which does not exist for the drive hardware.

Reactions: - NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check input configuration (machine data) and/or drive hardware. Power-up is interrupted.

Program Continuation: Switch control OFF - ON.

14400 Channel %1 block %2 tool radius compensation active at transformation switchover

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A change of transformation is not allowed when tool radius compensation is active.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Perform tool radius compensation in the NC part program with G40 (in a block with G00 or G01) before performing a transformation change.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14401 Channel %1 block %2 transformation not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The required transformation is not available.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Modify part program; program defined transformations only.
- Check MD 24100 TRAFO_TYPE_n (assigns the transformation to part program instructions).

Program Continuation: Clear alarm with the RESET key. Restart part program

14402 Channel %1 block %2 spline active at transformation change

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A change of transformation is not allowed in a spline curve section. A series of spline blocks must be concluded.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14403 Channel %1 block %2 preprocessing and main run might not be synchronized

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Positioning axis runs cannot be accurately calculated beforehand. Consequently, the position in the MCS is not known exactly. It might therefore be possible that a change in the multiple significance of the transformation has been performed in the main run although no provision was made for this in the preprocessing run.

Reactions: - Alarm display.

Remedy: Modify part program. Synchronize preprocessing run and main run.

Program Continuation: Clear alarm with the Delete key or NC START.

14404 Channel %1 block %2 illegal parameterization of transformation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Error has occurred when selecting transformation.

© Siemens AG, 2002. All rights reserved

Possible causes of error:
- An axis traversed by the transformation has not been enabled:
- is being used by another channel (-> enable)
- is in spindle mode (-> enable with SPOS)
- is in POSA mode (-> enable with WAITP)
- is concurrent Pos axis (-> enable with WAITP)
- Parameterization via machine data has an error
- Axis or geometry axis assignment to the transformation has an error,
- Machine data has an error (-> modify machine data, cold restart)

Note: Any axes that have not been enabled might be signaled via EXINAL_ILLEGAL_AXIS = 14092 or BSAL_SYSERRCHAN_RESET = 1011 instead of EXINAL_TRANSFORM_PARAMETER = 14404.

Transformation-dependent error causes can be in: TRAORI: -

TRANSMIT:
- The current machine axis position is unsuitable for selection (e.g. selection in the pole) (-> change position slightly).
- Parameterization via machine data has an error.
- Special requirement with respect to the machine axis has not been satisfied (e.g. rotary axis is not a modulo axis) (-> modify machine data, cold restart).

TRACYL:
- The programmed parameter is not allowed when transformation is selected.

TRAANG:
- The programmed parameter is not allowed when transformation is selected.
- Parameterization via machine data has an error.
- Parameter is faulty (e.g. TRAANG: Unfavorable angle value (-> modify machine data, cold restart)

Only with active "OEM transformation" compile cycle:
- The axes included in the transformation must be referenced!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify part program or machine data.

Only with active "OEM transformation" compile cycle:

Reference the axes included in the transformation before selecting transformation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14410 Channel %1 block %2 spline active at geometry axis changeover

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: It is not allowed to change the assignment of geometry axes to channel axes in a spline curve definition.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14411 Channel %1 block %2 tool radius compensation active at geometry axis changeover

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: It is not permissible to change the assignment of geometry axes to channel axes when tool radius compensation is active.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14412 Channel %1 block %2 transformation active at geometry axis changeover

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: It is not permissible to change the assignment of geometry axes to channel axes when transformation is active.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14413 Channel %1 block %2 fine tool correction: changeover geometry/channel axis not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: It is not permissible to change the assignment of geometry axes to channel axes during active tool fine compensation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14414 Channel %1 block %2 GEOAX function: incorrect call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The parameters for the GEOAX(...) call are incorrect. Possible causes:

- Uneven number of parameters.
- More than 6 parameters were specified.
- A geometry axis number was programmed which was smaller than 0 or greater than 3.
- A geometry number was programmed more than once.
- An axis identifier was programmed more than once.
- An attempt was made to assign a channel axis to a geometry axis which has the same name as one of the channel axes.
- An attempt was made to remove a geometry axis from the geometry axis grouping and the geometry axis has the same name as one of the channel axes.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program or correction block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14415 Channel %1 block %2 tangent control: changeover geometry/channel axis not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An assignment change of the geometry axes to channel axes is not permitted with active tangential control.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Change part program and delete active tangential control with TANGDEL.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14420 Channel %1 block %2 index axis %3 frame not allowed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis

Definitions: The axis is to be traversed as an indexing axis, but a frame is active. This is not allowed by machine data FRAME_OR_CORRPOS_NOTALLOWED.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify part program. Change machine data CORR_FOR_AXIS_NOT_ALLOWED.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14500 Channel %1 block %2 illegal DEF or PROC instruction in the part program

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: NC part programs with high-level language elements are divided into a preceding definition part followed by a program part. The transition is not marked specifically; a definition statement is not allowed to follow the first program command.

Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Put definition and PROFC statements at the beginning of the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14510 Channel %1 block %2 PROC instruction missing on subroutine call

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In subroutine calls with parameter transfer ("call-by-value" or "call-by-reference") the called subroutine must begin with a PROC statement.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Define the subroutine in accordance with the type used.

1. Conventional subroutine structure (without parameter transfer):

```
% SPF 123456
 :
M17
```

2. Subroutine structure with keyword and subroutine name (without parameter transfer):

```
PROC UPNAME
 :
M17
ENDPROC
```

3. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-value"):

```
PROC UPNAME (VARNAME1, VARNAME2, ...)
 :
M17
ENDPROC
```

4. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-reference"):

```
PROC UPNAME (Typ1 VARNAME1, Typ2 VARNAME2, ...)
 :
M17
ENDPROC
```

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14520 Channel %1 block %2 illegal PROC instruction in data definition section

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The PROC statement may only be programmed at the beginning of the subroutine.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify NC part program appropriately.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14530 Channel %1 block %2 EXTERN and PROC instruction do not correspond

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Subroutines with parameter transfer must be known before they are called in the program. If the subroutines are always available (fixed cycles) the control establishes the call interfaces at the time of system power-up. Otherwise an EXTERN statement must be programmed in the calling program.

Example:

```
N123 EXTERN UPNAME (TYP1, TYP2, TYP3, ...)
```

The type of the variable must match the type given in the definition (PROC statements) or it must be compatible with it. The name can be different.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Check the variable types in the EXTERN and the PROC statements for correspondence and correctness.

Program Continuation: Clear alarm with the RESET key. Restart part program

14600 Channel %1 block %2 reload buffer %3 cannot be established

Parameters: %1 = Channel number
%2 = Block number, label
%3 = File name

Definitions: The download buffer for "execute from external" could not be created. Possible causes:
- Not enough memory available (for minimum see MD $MN_MM_EXT_PROG_BUFFER_SIZE)
- No resources available for MMC NCK communication (see MD $MN_MM_EXT_PROG_NUM)
- The file already exists

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Release memory, e.g. by deleting part programs
- Modify MD $MN_MM_EXT_PROG_BUFFER_SIZE and/or $MN_MM_EXT_PROG_NUM.

Program Continuation: Clear alarm with the RESET key. Restart part program

14601 Channel %1 block %2 reload buffer could not be deleted

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The reload buffer for "execute from external" could not be deleted. Possible cause: MMC/ PLC communication was not terminated.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: All reload buffers are cleared on POWER ON.

Program Continuation: Clear alarm with the RESET key. Restart part program

14602 Channel %1 block %2 timeout during EXTCALL

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No connection could be established to the MMC for reloading of external subprograms (EXTCALL) within the time set in $MN_MMC_CMD_TIMEOUT.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Requirement: MMC102/103 with software version >= P4
- Check the connection to the MMC102/103
- Increase $MN_MMC_CMD_TIMEOUT.

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key. Restart part program

14610 Channel %1 block %2 compensation block not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An alarm was output which could be eliminated basically via program correction. Since the error occurred in a program which is processed from external, a compensation block/ program correction is not possible.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Abort program with reset.
- Correct program on MMC or PC.
- Restart reloading (possibly with block search and interrupt location).

Program Continuation: Clear alarm with the RESET key. Restart part program

14650 Channel %1 block %2 SETINT instruction with invalid ASUP input

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Asynchronous subroutines are subroutines that are executed following a hardware input (interrupt routine started by a rapid NCK input).
The NCK input number must lie between 1 and 8. It is assigned a priority from 0 to 128 (1 is the highest priority) in the SETINT instruction with the keyword PRIO = ...
Example:
If NCK input 5 changes to "1" the subroutine LIFT_Z should be started with the highest priority.
N100 SETINT (5) PRIO = 1 LIFT_Z

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the NCK input of the SETINT statement with a value of not less than 1 or greater than 8.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14660 Channel %1 block %2 SETINT instruction with invalid priority

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The NCK input number must lie between 1 and 8. It is assigned a priority from 0 to 128 (1 is the highest priority) in the SETINT instruction with the keyword PRIO = ...
Example:
If NCK input 5 changes to "1" the subroutine LIFT_Z should be started with the highest priority.
N100 SETINT (5) PRIO = 1 LIFT_Z

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the priority of the NCK input with a value of not less than 1 or greater than 128.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14700 Channel %1 block %2 timeout during command to interpreter

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A timeout has occurred in control-internal commands such as ANWAHL (part program selection), RESET (channel reset), REORG (reorganization of the preprocessing buffer) and NEWCONFIG (change in the configuration-specific machine data = warm restart).

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. If the runtime error occurred as the result of a temporary excessive load on the system (e.g. in the MMC area or in OEM application) error-free execution is possible on repeating the program or operator action. Otherwise, the A&D MC system support should be contacted with a precise description of the error situation:
(contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.:see alarm 1000)

Program Continuation: Switch control OFF - ON.

14701 Channel %1 block %2 number of available NC blocks reduced by %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of non-available blocks

Definitions: After reset, it has been found that the number of available blocks has decreased compared with the last reset. This is due to a system error. Part program execution can be resumed after the alarm has been acknowledged. If the number of blocks no longer available is less than 28060 MM_IPO_BUFFER_SIZE, then the POWERON alarm 14700 is output.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Proceed as in the case of a system error.

Program Continuation: Clear alarm with the RESET key. Restart part program

14710 Channel %1 block %2 error in initialization sequence in function %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Identifier of the function which caused the error

Definitions: Initialization blocks are generated (or not) after control power-up, program reset and program start, depending on the settings in machine data $MC_RESET_MODE_MASK and $MC_START_MODE_MASK.
Errors can occur because of incorrect machine data settings. The errors are output with the same error messages as appear if the function was incorrectly programmed in the part program.
This alarm is generated in addition, in order to indicate that an error refers to the initialization sequence.
Parameter %3 specifies which function triggers the alarm:
Control power-up and (program) RESET:

© Siemens AG, 2002. All rights reserved

Value:
0: Error during synchronization preprocessing/main run
1: Error on selection of tool length compensation
2: Error on selection of transformation
3: Error on selection of zero offset
The macro definitions and cycle interfaces are also read in during the power-up procedure. If an error occurs here, this is indicated by value = 4, or value = 5
(Program) START:
Value
100: Error during synchronization preprocessing/main run
101: Error on selection of tool length compensation
102: Error on selection of transformation
103: Error on selection of synchronized spindle
104: Error on selection of zero offset
Particularly when tool management is active, it is possible that a tool on the spindle or the toolholder is disabled but still needs to be activated.
These tools are automatically activated on RESET. On START, machine data $MC_TOOL_CHANGE_ERROR_MODE can be used to specify whether an alarm is generated or an automatic bypass strategy is applied.
If the parameter contains 3 values from 200 to 203, this means that an insufficient number of NC blocks is available for NC block preparation on certain commands (ASUP start, overstore selection, teach-in).
Remedy: Increase machine data $MC_MM_NUM_BLOCKS_IN_PREP.

Reactions:
- Interpreter stop
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Please inform the authorized personnel/service department.
On parameter %3= 0 -3:
If the alarm or alarms occur on RESET:
Check the settings of machine data $MC_RESET_MODE_MASK,
$MC_TOOL_RESET_VALUE, $MC_TOOL_PRESEL_RESET_VALUE,
$MC_TOOL_RESET_NAME (only if tool management is active),
$MC_CUTTING_EDGE_RESET_VALUE, $MC_SUMCORR_RESET_VALUE,
$MC_TOOL_CARRIER_RESET_VALUE,
$MC_GCODE_RESET_VALUES, $MC_EXTERN_GCODE_RESET_VALUES,
$MC_TRAFO_RESET_VALUE,
$MC_COUPLE_RESET_MODE_1,
$MC_CHBFRAME_RESET_MASK
On parameter %3= 100 - 104:
Check the setting of machine data $MC_START_MODE_MASK and the machine data specified under '..._RESET_...'. Check machine data. If tool management is active, remove the specified tool from the toolholder/spindle and, if necessary, cancel the 'disabled' status.
On parameter %3= 4 or 5:
Check macro definitions in _N_DEF_DIR
Check cycle directories _N_CST_DIR and _N_CUS_DIR
On parameter %3= 200 to 203:
Increase machine data $MC_MM_NUM_BLOCKS_IN_PREP.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

14720 Channel %1 block %2 axes for centerless transformation not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the channel not all of the axes/spindles are available that have been defined in machine data for centerless grinding.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
1. Modify part program.
2. Modify machine data:
24110 TRAFO_AXES_IN_n
21522 TRACLG_GRINDSPI_NR
21524 TRACLG_CTRLSPI_NR.

Program Continuation: Clear alarm with the RESET key. Restart part program

14730 Channel %1 block %2 conflict at activation of centerless transformation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:
- Centerless transformation may not be activated when:
- G96 is active and regulating spindle is also master spindle.
- Regulating spindle is in interdependent grouping.
- Axes of centerless transformation overlap with an active transformation and a tool is active.
- For grinding or for regulating wheel spindle, tools are active that are not centerless tools (T1, T2).
- Constant wheel peripheral speed for the regulating spindle is active.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Modify part program.
- Check tool data.
- Check machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

14740 Channel %1 block %2 no tool data available for centerless grinding

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: For centerless grinding, the tool data must be contained in T1, D1 (grinding wheel) or T2,D1 (regulating wheel). An error has been found here.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Modify part program.
- Check tool data.
- Check machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

14745 Channel %1 block %2 centerless grinding not active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt has been made to switch off the centerless grinding function even though it was not active.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

14750 Channel %1 block %2 too many auxiliary functions programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: More than 10 auxiliary functions have been programmed in an NC block.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check whether all auxiliary functions are necessary in one block - modal functions need not be repeated. Create separate auxiliary function block or divide the auxiliary functions over several blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14751 Channel %1 block %2 resources for motion synchronous actions not sufficient (code: %3)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Identifier

Definitions: To process motion synchronous actions resources are required. They are configured via the machine data $MC_MM_IPO_BUFFER_SIZE, $MC_MM_NUM_BLOCKS_IN_PREP and $MC_MM_NUM_SYNC_ELEMENTS. If these resources are insufficient for executing the part program, then this alarm is issued. The parameter %3 shows which resource has run out:
Increase identifier <= 2: $MC_MM_IPO_BUFFER_SIZE or
$MC_MM_NUM_BLOCKS_IN_PREP.
Increase identifier > 2: $MC_MM_NUM_SYNC_ELEMENTS.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct part program or increase resources.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14752 Channel %1 block %2 DELDTG | STOPREOF conflict

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In a block of motion synchronous actions referring to a motion block, both DELDTG (delete distance-to-go) and STOPREOF (preprocessing stop) have been programmed.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: The functions DELDTG and STOPREOF exclude each other in a block.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14753 Channel %1 block %2 motion synchronous actions with illegal interpolation type

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The active interpolation type (e.g. 5-axis interpolation) is not allowed for the motion synchronous action or for the function "Several feeds".
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14754 Channel %1 block %2 motion synchronous actions and wrong feed type

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The active feed type is not allowed for the motion synchronous action or for the function "Several feeds".
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14755 Channel %1 block %2 motion synchronous actions without traverse motion

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed motion synchronous action and the function "Several feeds" require a traversing motion or the value of the traversing motion is 0.
This alarm is no longer used after P3.2.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14756 Channel %1 block %2 motion synchronous action and wrong value

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Value of the synchronous action or the function "Several feeds" is not allowed.
Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.
Remedy: Modify part program. Check whether a negative value was entered for a synchronous action.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14757 Channel %1 block %2 motion synchronous action and wrong type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Programmed combination between action and type of motion synchronous action is not allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14758 Channel %1 block %2 programmed value not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The synchronous variables $AA_LOAD, $AA_TORQUE, $AA_POWER and $AA_CURR are available only for the 611D drive. They are activated by the machine data MDC 36730 DRIVE_SIGNAL_TRACKING. The system variable $VA_IS: Safe Actual Position is available only if the machine data $MA_SAFE_FUNCTION_ENABLE has been set and the option $ON_NUM_SAFE_AXES has been set to a sufficient size.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify program or machine data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14759 Channel %1 block %2 motion synchronous action and wrong axis type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When there are several feeds, a spark-out time, or a retraction stroke for path motions, at least one GEO axis must be programmed. If the block also contains synchronous axes and there are several feeds, the feedrate for the synchronous axes is matched implicitly. No retraction stroke takes place for synchronous axes. However, after retraction stroke or spark-out time, the distance-to-go is also deleted in the block for the synchronous axes.
The alarm is no longer used on P3.2.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the axis as positioning axis with axial feed, return stroke or spark-out time.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14760 Channel %1 block %2 auxiliary function of a group programmed repeatedly

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The M and H functions can be divided up as required over machine data in groups in any variation. Auxiliary functions are thus put into groups that mutually preclude several individual functions of one group. Within one group only one auxiliary function is advisable and permissible.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Only program one help function per help function group. (For the group division, see the machine manufacturer's programming guide).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14761 Channel %1 block %2 motion synchronous action: DELDTG function not allowed with active tool radius compensation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Rapid delete distance-to-go for synchronous actions is not allowed with DELDTG when tool radius compensation is active.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Deactivate tool radius compensation before performing rapid delete distance-to-go and then reselect

or

- as of SW 4.3: "Delete distance-to-go without preparation".

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14762 Channel %1 block %2 too many PLC variables programmed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The number of programmed PLC variables auxiliary functions has exceeded the maximum permissible number. The number is set in MD 28150 $MC_MM_NUM_VDIVAR_ELEMENTS.

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program or machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

14763 Channel %1 block %2 too many link variables programmed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The number of NCU link variables programmed exceeds the maximum limit. The number is defined in MD $MC_MM_NUM_LINKVAR_ELEMENTS.

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program or machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

14764 NCU link cannot transfer all link variables immediately

Definitions: Informational alarm for NC program developer.

A value assignment to a link variable (e.g. $a_dld[16]=19) is performed in the main run and transferred via NCU link to all NCUs in the link network. The bandwidth of this connection restricts the number of value assignments which can be transferred in one interpolation cycle.

All value assignments are combined in the next main run block and performed immediately this block is executed. A main run block is the block at which you would stop in single block mode SLB1.

Examples:

Blocks with a real traversing movement (G0 X100), Stopre, G4, WAITM, WAITE,...

The alarm occurs if the number of link variables set in any interpolation cycle exceeds the number that can be transferred. The link variables are not transferred until one of the next interpolation cycles. The assignment is not lost!

Reactions: - Alarm display.
- Warning display.

Remedy: Insert main run blocks between the assignments if the program sequence allows. See also $A_LINK_TRANS_RATE.

Program Continuation: Clear alarm with the Delete key or NC START.

14765 NCU link cannot transfer all link variables

Definitions: A value assignment to a link variable (e.g. $a_dld[16]=19) is performed in the main run and transferred via NCU link to all NCUs in the link network. The bandwidth of this connection restricts the number of value assignments which can be transferred in one interpolation cycle. Assignment operations which are not transferred are stored in a buffer memory. This buffer is full!

All value assignments are combined in the next main run block and performed immediately this block is executed.

A main run block is the block at which you would stop in single block mode SLB1.

Examples: Blocks with a real traversing movement (G0 X100), Stopre, G4, WAITM, WAITE,...

Link variable scanning operations are not affected (e.g.: R100= $a_dld[16])

Reactions: - Alarm display.
- NC Stop on alarm.
- NC Start disable in this channel.
- Interface signals are set.

Remedy: Insert main run blocks which require a sufficient number of interpolation cycles for execution (e.g. G4 F10) between the assignments. A block with an additional preprocessor stop does not improve the situation! See also $A_LINK_TRANS_RATE, for a variable which you can test before an assignment.

Program Continuation: Clear alarm with the RESET key. Restart part program

14766 NCU link is heavily loaded, impending memory shortage

Definitions: Informational alarm for NC program developer.

The capacity of the NCU link is not large enough to transfer all the data. This non-cyclic data includes link variable assignments, machine data write operations, values for container switches and setting data write operations.

This type of data is buffered and is not lost. The buffer memory is now 70% full.

Reactions: - Alarm display.
- Warning display.

Remedy: The timing of cyclic data should not be distorted in the NC program.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

14767 Machine data matching via NCU link not complete

Definitions: A non-released option has been used in the block.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Change less setting or machine data at the same time.

Program Continuation: Switch control OFF - ON.

14770 Channel %1 block %2 auxiliary function programmed incorrectly

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The permissible number of programmed auxiliary functions per NC block has been exceeded or more than one auxiliary function of the same auxiliary function group has been programmed (M and S function).

In the user-defined auxiliary functions, the maximum number of auxiliary functions per group in the NCK system settings has been defined for all auxiliary functions by means of the machine data 11100 AUXFU_MAXNUM_GROUP_ASSIGN (default: 1)

For each user-defined auxiliary function to be assigned to a group, the assignment is effected through 4 channel-specific machine data.

Return jump from asynchronous subprogram with M02/M17/M30, whereby the M code is not alone in the block. This is impermissible if the asynchronous subprogram interrupts a block with WAITE, WAITM or WAITMC. Remedy: Program M02/M17/M30 alone in the block or replace via RET.

22010 AUXFU_ASSIGN_TYPE: type of auxiliary function, e.g. M

22000 AUXFU_ASSIGN_GROUP: required group

22020 AUXFU_ASSIGN_EXTENSION: any required extension

22030 AUXFU_ASSIGN_VALUE: function value

Reactions:
- Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Correct the part program - max. 16 auxiliary functions, max. 5 M functions per NC block, max. 1 auxiliary function per group.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14780 Channel %1 block %2 unreleased option used

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: A non-released option has been used in the block.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program, retrofit option.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14790 Channel %1 block %2 axis %3 programmed by PLC

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Axis

© Siemens AG, 2002. All rights reserved

Definitions: In the NC block, an axis has been programmed that is already being traversed by the PLC.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Modify part program, do not use this axis.
- Stop traversing motion of the axis by the PLC, modify part program (insert WAITP).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14800 Channel %1 block %2 programmed path speed less or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A negative F value has been programmed in conjunction with the G functions G93, G94, G95 or G96. The path velocity may be programmed in the range 0.001 to 999 999.999 [mm/min, mm/rev, deg/min, deg/rev] for the metric input system and 0.000 1 to 39 999.999 9 [inch/min, inch/rev] for the inch input system.

Reactions:
- Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Program the path velocity (geometric sum of the velocity components of the geometry axes involved) within the limits given above.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14810 Channel %1 block %2 negative axis speed programmed for positioning axis %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis

Definitions: A negative feed (FA value) has been programmed for the displayed axis presently operating as a positioning axis. The positioning velocity may be programmed in the range 0.001 to 999 999.999 [mm/min, deg/min] for the metric input system and 0.000 1 to 39 999.999 9 [inch/min, inch/rev] for the inch input system.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the positioning velocity within the limits given above.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14811 Channel %1 block %2 incorrect value range for acceleration of axis/spindle %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis, spindle

Definitions: A value outside of the permissible input range of the programmed acceleration has been used. Values of between 1 and 200 % are possible.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Adjust the value range in accordance with the Programming Guide. Values of 1 ... 200% are allowed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14812 Channel %1 block %2 SOFTA not available for axis %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis

Definitions: SOFT is to be set as type of motion control for an axis. This is not possible because a bent acceleration characteristic has been selected for this axis via machine data.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program or machine data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14815 Channel %1 block %2 negative thread lead change programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A negative thread lead change has been programmed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the value assignment. The programmed F value should be greater than zero. Zero is allowed but has no effect.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14820 Channel %1 block %2 negative value for maximum spindle speed programmed with constant cutting speed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: For the function "Constant cutting speed G96" a maximum spindle speed can be programmed with the keyword LIMS=.... The values are in the range 0.1 - 999 999.9 [rev/min].

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program the maximum spindle speed for the constant cutting speed within the limits given above. The keyword LIMS is modal and can either be placed in front of or within the block that selects the constant cutting speed.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14821 Channel %1 block %2 error in selection or deselection of GWPS

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: On selecting grinding wheel surface speed programming with GWPSON, one of the following errors occurred:

- An attempt has been made to select the GWPS programming for a spindle that has already been assigned to another tool by TMON, GWPSON, CLGON or activation of the tool length compensation.
- Selection does not refer to a grinding-specific tool (400-499).
- An attempt has been made to select GWPS for the active tool although the TLC is not switched on.

© Siemens AG, 2002. All rights reserved

- Selection refers to an invalid spindle number.
- A grinding wheel radius equal to zero was specified.

On deselecting grinding wheel surface speed programming with GWPSOFF, one of the following errors occurred:

- Deselection does not refer to a grinding-specific tool (400-499).
- An attempt has been made to deselect GWPS for the active tool although the tool length compensation has not been activated.
- Selection refers to an invalid spindle number.

Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy:
- Check GWPSON and GWPSOF command.
- Check tool compensation data:

$TC_DP1 : 400 - 499;

$TC_TGP1: Spindle number.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14822 Channel %1 block %2 incorrect programming of GWPS

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: When selecting GWPS (constant grinding wheel peripheral speed) with GWPSON or programming the GWPS with "S[spindle number] = value" one of the following errors has occurred:

Invalid spindle number.

Invalid parameter number for radius calculation in $TC_TPG9.

The following values are valid:

3 for $TC_DP3 (length 1)

4 for $TC_DP4 (length 2)

5 for $TC_DP5 (length 3)

6 for $TC_DP6 (radius)

Invalid angle in $TC_TPG8.

The following values are valid: -90 <= $TC_TPG8 < +90.

A grinding wheel radius equal to zero was specified.

Reactions: - Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Check tool compensation data.
- $TC_DP1 : 400 - 499.
- $TC_TPG1: Spindle number.
- $TC_TPG8: Inclination angle for slope grinding wheel.
- $TC_TPG9: Compensation parameters for radius computation, e.g. 3 for $TC_GP3.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14823 Channel %1 block %2 error on selection or deselection of tool monitoring

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: On selecting tool monitoring with TMON, one of the following errors occurred:
- Selection does not refer to a grinding-specific tool (400-499).
- Selection refers to an invalid spindle number.

© Siemens AG, 2002. All rights reserved

- An attempt has been made to select tool monitoring for a spindle that is already assigned to another tool by TMON, GWPSON, CLGON or activation of tool length compensation.
- An attempt has been made to select tool monitoring for the active tool although no tool length compensation has been activated.
- Invalid parameter number for radius computation in $TC_TPG9.

The following values are valid:

3 for $TC_DP3 (length 1)

4 for $TC_DP4 (length 2)

5 for $TC_DP5 (length 3)

6 for $TC_DP6 (radius)

- A grinding wheel radius equal to zero was specified.

On deselecting tool monitoring with TMOF, one of the following errors occurred:

- Deselection does not refer to a grinding-specific tool (400-499).
- An attempt has been made to deselect tool monitoring for the active tool although tool length compensation is not active.
- The deselection refers to an invalid spindle number.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check TMON and TMOF command.

Check tool compensation data.

- $TC_DP1 : 400 - 499.
- $TC_TPG1: Spindle number.
- $TC_TPG8: Inclination angle for slope grinding wheel.
- $TC_TPG9: Parameter number for radius computation, e.g. 3 for $TC_GP3.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14824 Channel %1 block %2 conflict with GWPS

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The functions of constant grinding wheel surface speed GWPS and constant cutting speed G96 S... have been activated at the same time for a spindle.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14840 Channel %1 block %2 incorrect value range for constant cutting speed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The programmed cutting speed is not within the input range

Input range metric: 0.01 to 9 999.99 [m/min]

Input range inch: 0.1 to 99 999.99 [inch/min].

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program cutting speed under address S within the permissible range of values.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

14900 Channel %1 block %2 center point and end point programmed simultaneously

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When programming a circle by means of the opening angle, the circle center point was programmed together with the circle end point. This is too much information for the circle. Only one of the two points is allowed.

Reactions:
- Correction block is reorganized.
- Interface signals are set.
- Alarm display.

Remedy: Select the programming variant guaranteeing that the dimensions are definitely taken over from the workpiece drawing (avoidance of calculation errors).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14910 Channel %1 block %2 invalid angle of aperture for programmed circle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When programming a circle by means of the opening angle, a negative opening angle or an opening angle greater than or equal to 360 degrees has been programmed.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program opening angle within the allowed range of values between 0.0001 and 359.9999 [degrees].

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14920 Channel %1 block %2 intermediate point of circle incorrect

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When programming a circle by means of an intermediate point (CIP) all 3 points (initial, end and intermediate points) are on a straight line and the intermediate point (programmed by means of interpolation parameters I, J, K) is not located between the initial and end points.

If the circle is the component of a helix, the specified number of turns (keyword TURN=...) determines further block processing:

- TURN>0: alarm display because the circle radius is infinitely great.
- TURN=0 and CIP specified between initial and end points. A straight line is generated between the initial and end points (without alarm message).

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Locate the position of the intermediate point with the parameters I, J and K in such a way that it actually is located between the initial and end points of the circle or do not make use of this type of circle programming and instead program the circle with radius or opening angle or center point parameters.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15000 Channel %1 block %2 channel-sync instruction using illegal mark

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: A WAITM/WAITMC/SETM/CLEARM instruction was programmed with a marker number that was less than 1 or greater (MAXNUM_MARKER * MAXNUM_CHANNELS).
Exception: CLEARM(0) is allowed and clears all markers in channel!
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Correct the instruction accordingly.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15010 Channel %1 block %2 program coordination instruction with invalid channel number

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A WAITM, WAITMC, INIT or START instruction was programmed with an invalid channel number.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Correct the instruction accordingly.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15020 Channel %1 block %2 CHANDATA instruction cannot be executed. Channel %3 is not active

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String (CHANDATA parameter)
Definitions: With a CHANDATA instruction, the data input for a channel is selected that has not been activated. For structural reasons, the input of multi-channel data must take place twice.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Please inform the authorized personnel/service department.
• Activate the channel concerned by means of machine data or option data or
• Cancel the CHANDATA instruction and all following assignments to channel data. This error message occurs regularly when first reading in an INITIAL Init block with which a multi-channel system is to be installed. In this case:
1. NCK Restart must be executed in order to activate the global machine data already input for the installation of the other channels.
2. Input of the INITIAL Init block must be repeated.
Program Continuation: Switch control OFF - ON.

15021 Channel %1 block %2 CHANDATA instruction with invalid channel number

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A CHANDATA instruction is used to enter data for an illegal channel, e. g. <1,> maximum number of channels, not the active channel.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Program CHANDATA instruction in accordance with the actual configuration.
Program Continuation: Clear alarm with the RESET key. Restart part program

15025 CHANDATA(%2): channel is not active. Channel data will be ignored.

Parameters: %1 = Channel number
%2 = CHANDATA parameter
Definitions: With a CHANDATA instruction, the data input for a channel is selected that has not been activated.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
Remedy: This is an informational alarm referring to the fact that the file loaded into the NCK contains data of an inactive channel. The number of the inactive channel is specified. Subsequently, the data of this channel are not available in the NCK.
The alarm may have two causes:
(1.) The channel is supposed to be activated by a following NCK RESET/POWER ON, i.e. the file must subsequently be reloaded.If the alarm occurs again, the reason is: (2) the specified channel is actually not supposed to be activated, however, the file contains the relevant data.
For the second reason, please check whether the system has correctly not activated the channel mentioned.
If the channel has been activated, operation may be continued after another NCK RESET/POWER ON without further measures, i.e. reloading the file is not required. If the channel has not been activated, make sure that the channel inactivated by mistake is re-activated.
If the settings of the channel activation are part of the file to be loaded (e.g. archive file), the file must either be modified with the relevant program or the file has to be created once more in the same system with the correct channel number.
Similar alarms: 15020, 15021.
Program Continuation: Switch control OFF - ON.

15030 Channel %1 block %2 different measurement system settings

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The INCH or METRIC instruction describes the system of measurement in which the data blocks have been read from the control. In order to prevent the incorrect interpretation of data intended for a particular system of measurement, a data block is only accepted if the above instruction matches the active system of measurement.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Change the system of measurement or load a data block which matches the system of measurement.
Program Continuation: Clear alarm with the RESET key. Restart part program

15100 Channel %1 block %2 REORG abort caused by log file overflow

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In order to synchronize the preprocessing run and the main run with REORG, the control accesses modification data which are maintained in a logfile. The alarm indicates that no more capacity is available in the logfile for the specified block in the channel.

© Siemens AG, 2002. All rights reserved

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Please inform the authorized personnel/service department. No remedial measures are available for the further execution of the current NC program, however:

1. Reduce log file size requirement by:

Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops STOPRE.

2. The logfile should be increased in size by means of the channel-specific data:

Modify MD 28000: MM_REORG_LOG_FILE_MEM and

Modify MD 28010: MM_REORG_LUD_MODULES

Notice!

A change in these machine data also causes a reallocation of the NCK user memory and the standard machine data are then loaded. Unless a data save is performed, there will be a LOSS OF DATA!'

Program Continuation: Clear alarm with the RESET key. Restart part program

15110 Channel %1 block %2 REORG not possible

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: In order to synchronize the preprocessing run and the main run with REORG, the control accesses modification data which are maintained in a logfile. The alarm indicates that no more capacity is available in the logfile for the specified block in the channel.

The alarm message means that the logfile has been deleted in order to obtain additional memory for program reorganization. Consequently, it is no longer possible to REORG the preprocessing memory up to the next coincidence point.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. No remedial measures are available for the further execution of the current NC program, however:

1. Reduce log file size requirement by:

Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops

STOPRE.

2. The logfile should be increased in size by means of the channel-specific data:

Modify MD 28000: MM_REORG_LOG_FILE_MEM and

Modify MD 28010: MM_REORG_LUD_MODULES

Notice!

A change in these machine data also causes a reallocation of the NCK user memory and the standard machine data are then loaded. Unless a data save is performed, there will be a LOSS OF DATA!

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

15150 Channel %1 block %2 reload from external aborted

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Execution from external was aborted because the reload buffer does not have enough machine function blocks (traversing blocks, auxiliary function, dwell time etc.). Background: When already executed machine function blocks are released, memory becomes available in the reload buffer. If machine function blocks are no longer released, nothing can be reloaded - this results in a deadlock situation.

Example: Definition of extremely long curve tables via execution from external.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Insert machine function blocks in the part program.
- Increase the size of the reload buffer ($MN_MM_EXT_PROG_BUFFER_SIZE).
- Decrease the size of the curve table (Note: Blocks within CTADDEF/CTABEND are not machine function blocks).

Program Continuation: Clear alarm with the RESET key. Restart part program

15160 Channel %1 block %2 wrong preprocessing configuration

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The following deadlock has been found in the interpreter: A block element is needed but the block element memory is empty and there is no likelihood of getting new block elements by processing the preprocessing/main run queue because this queue itself is empty.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Modify the block search configuration in machine data 28060 MM_IPO_BUFFER_SIZE (decrease size of IPO buffer if necessary) or 28070 MM_NUM_BLOCKS_IN_PREP.

Program Continuation: Clear alarm with the RESET key. Restart part program

15165 Channel %1 block %2 error when translating or interpreting PLC Asup %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String

Definitions: At part program start, the data part of the selected PLC Asup is conditioned. If an error occurs (translator or interpreter) first this alarm is output and then a translator or interpreter alarm that describes the error in more detail. The error causes the interpreter to stop. Correction block is not possible.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

15166 Channel %1 user system asup _N_ASUP_SPF not available

Parameters: %1 = Channel number

Definitions: By means of the machine data 11610 $MN_ASUP_EDITABLE the function "User-defined system asup" has been activated. However, the relevant user program could not be found in the specified search path:
- 1. /_N_CUS_DIR/_N_ASUP_SPF
- 2. /_N_CUS_DIR/_N_ASUP_SPF

The default system asups are used.

Reactions: - Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Load the user-defined system asup in /_N_CUS_DIR/_N_ASUP_SPF or /_N_CMA_DIR/_N_ASUP_SPF laden.
Program Continuation: Clear alarm with the RESET key. Restart part program

15170 Channel %1 block %2 program %3 could not be compiled

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String
Definitions: An error has occurred in compile mode. The (compiler) error message refers to the program specified here.
Reactions: - Alarm display.
Remedy: Modify part program.
Program Continuation: Clear alarm with the Delete key or NC START.

15171 Channel %1 block %2 compiled program %3 older than the relevant subroutine

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Compiled program file name
Definitions: When calling a precompiled subroutine, it was noticed that the compiled program is older than the relevant SPF file. The compiled program was deleted and during start the subroutine is executed instead of the compiled program.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Perform another precompilation.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15175 Channel %1 block %2 program %3. Interfaces could not be built

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String
Definitions: An error has occurred in interface generation mode. The (compiler) error message refers to the program specified here. In particular when loading new cycle program on the NCK, problems can occur if the value settings in machine data $MN_MM_NUM_MAX_FUNC_NAMES and $MN_MM_NUM_MAX_FUNC_PARAM are too small.
Reactions: - Alarm display.
Remedy:
- Modify part program.
- If new cycle programs have been loaded on the NCK, you will normally need to increase the values of $MN_MM_NUM_MAX_FUNC_NAMES and $MN_MM_NUM_MAX_FUNC_PARAM. See also the explanations for alarm 6010.

Program Continuation: Clear alarm with the Delete key or NC START.

15180 Channel %1 block %2 program %3 cannot be executed as INI file

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String
Definitions: Errors were found when processing an initialization program (INI file), or a GUD or macro definition file (DEF file).
The error message which is then displayed refers to the program specified here.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
Remedy: Correct the initialization program (INI file), or the GUD or macro definition file (DEF file).
In connection with Alarm 12380 or 12460, also change the memory configuration.
Program Continuation: Clear alarm with the Delete key or NC START.

15185 Channel %1 %2 errors in INI file

Parameters: %1 = Channel number
%2 = Number of detected errors
Definitions: Errors were found when processing initialization programs (INI files), GUD or macro definition files (DEF files).
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
Remedy: Please inform the authorized personnel/service department. Correct the INI or DEF file or correct the MD and create a new INI file (via "Upload").
Program Continuation: Switch control OFF - ON.

15190 Channel %1 block %2 not enough free memory for subroutine call

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The following deadlock has been found in the interpreter: Memory is needed for calling a subroutine. The module memory is, however, empty and there is no prospect of module memory becoming free again by executing the preprocessing/main run queue, because this queue is empty.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Increase machine data 28010 MM_NUM_REORG_LUD_MODULES/28040 MM_LUD_VALUES_MEM / 18210 MM_USER_MEM_DYNAMIC or program a preprocessing stop STOPRE before calling the subroutine.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15300 Channel %1 block %2 invalid number-of-passed-blocks during block search

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the function "Block search with calculation" a negative number of passes has been entered in column P (number of passes). The permissible range of values is P 1 - P 9 999.
Reactions: - Alarm display.
Remedy: Enter only positive number of passes within the range of values.
Program Continuation: Clear alarm with the Delete key or NC START.

15310 Channel %1 block %2 file requested during block search is not available

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: During block search, a target has been specified with a program that has not been loaded.
Reactions: - Alarm display.
Remedy: Correct the specified search target accordingly or reload the file.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

15320 Channel %1 block %2 invalid block search command

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The block search command (type of search target) is smaller than 1 or greater than 5. It is entered in column type of the block search window. The following block search orders are allowed.

Type	Meaning
1	Search for block number
2	Search for label
3	Search for string
4	Search for program name
5	Search for line number in a file

Reactions: - Alarm display.

Remedy: Modify the block search command.

Program Continuation: Clear alarm with the Delete key or NC START.

15330 Channel %1 block %2 invalid block number as search target

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Syntax error! Positive integers are allowed as block numbers. Block numbers must be preceded by ":" and subblocks by an "N".

Reactions: - Alarm display.

Remedy: Repeat the input with corrected block number.

Program Continuation: Clear alarm with the Delete key or NC START.

15340 Channel %1 block %2 invalid label as search target

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Syntax error! A label must have at least 2 but no more than 32 characters, and the first two characters must be alphabetic or underscore characters. Labels must be concluded with a colon.

Reactions: - Alarm display.

Remedy: Repeat the input with corrected label.

Program Continuation: Clear alarm with the Delete key or NC START.

15350 Channel %1 block %2 search target not found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The specified program has been searched to the end of the program without the selected search target having been found.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Check the part program, change the block search (typing error in the part program) and restart the search.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

15360 Channel %1 illegal target of block search (syntax error)

Parameters: %1 = Channel number
Definitions: The specified search target (block number, label or string) is not allowed in block search.
Reactions: - Alarm display.
Remedy: Correct target of block search.
Program Continuation: Clear alarm with the Delete key or NC START.

15370 Channel %1 target of block search not found

Parameters: %1 = Channel number
Definitions: In a block search, an impermissible search target has been specified (e.g. negative block number).
Reactions: - Alarm display.
Remedy: Check the specified block number, label or character string. Repeat entry with correct search target.
Program Continuation: Clear alarm with the Delete key or NC START.

15380 Channel %1 block %2 illegal incremental programming in axis %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis
Definitions: The first axis programming after "search to block end point" is performed incrementally. This is not allowed in the following situations:
- After searching the target a transformation change has taken place.
- A frame with rotation component is active. The programmed axis is involved in the rotation.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy:
- Find search destination in which the axes are programmed using an absolute reference.
- Deactivate adding of the accumulated search position with $SC_TARGET_BLOCK_INCR_PROG = FALSE.
- Use search run with calculation "at contour".

Program Continuation: Clear alarm with the RESET key. Restart part program

15390 Channel %1 block %2 %3 not executed during block search

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source symbol
Definitions: During block search, commands for switching, deleting and defining of the electronic gear are not executed and not gathered but simply skipped.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Set the desired gear status via asynchronous subprogram.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

15395 Channel %1 master-slave not executable during block search

Parameters: %1 = Channel number

Definitions: A master-slave coupling is to be closed in the part program via the instruction MASLON. The position offset $P_SEARCH_MASLD, however, cannot be correctly calculated during block search, as the axes to be coupled are located in different channels.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Make sure that all relevant axes are in the same channel.

Program Continuation: Clear alarm with the RESET key. Restart part program

15400 Channel %1 block %2 selected initial init file does not exist

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The operator has selected an INI block for a read, write or execution function which:

1. Does not exist in the NCK range or

2. Does not have the necessary protection level required for performing the function.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Check whether the selected INI block is contained in the file system of the NCK. The present protection level must be selected to be at least equal to (or greater than) the protection level that has been defined for the read, write or execution function at the time of creating the file.

Program Continuation: Clear alarm with the RESET key. Restart part program

15410 Channel %1 block %2 initialization file contains invalid M function

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The only M function allowed in an Init block is the M02, M17 or M30 end-of-program function.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Remove all M functions from the Init block except for the end identifier.

An Init block may contain value assignments only (and global data definitions if they are not defined again in a program that can be executed later) but no motion or synchronous actions.

Program Continuation: Clear alarm with the RESET key. Restart part program

15420 Channel %1 block %2 instruction in current mode not allowed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The alarm is output in the following situation:

- The interpreter has detected an illegal instruction (e.g. a motion command) while processing an INI file or a definition file (macro or GUD).

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy:
- Correct the INI, GUD or macro file.
- Correct part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

15450 Channel %1 block %2 compiled program cannot be stored

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the compile mode, a compiled program could not be stored. One of the following reasons applies:
- Not enough memory
- Intermediate code line (compilate) too large

Reactions: - Alarm display.

Remedy: Create space in work memory or modify part program (make it less complex).

Program Continuation: Clear alarm with the Delete key or NC START.

15460 Channel %1 block %2 syntax error with modal function

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The addresses programmed in the block are not compatible with the modal syntax-determining G function.
Example:
N100 G01 ... I .. J.. K.. LF

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the displayed block and ensure that the G functions and addresses in the block are in agreement.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15500 Channel %1 block %2 illegal angle of shear

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The function CSHEAR has been called with an illegal (impossible) angle of shear, e.g. when the sum of angles between the axis vectors is greater than 360 degrees.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Program the angle of shear in accordance with the geometrical conditions of the machine and workpiece system.

Program Continuation: Clear alarm with the RESET key. Restart part program

15700 Channel %1 block %2 illegal cycle alarm number %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Cycle alarm number

Definitions: A SETAL command has been programmed with a cycle alarm number less than 60 000 or greater than 67 999.

© Siemens AG, 2002. All rights reserved

Alarm reactions of Siemens standard cycles:
Nos. 61,000 -61,999: Interpreter stop; delete with Reset
Nos. 62 000 - 62 999: Compensation block; delete with NC Start

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Program alarm number in the SETAL instruction in the correct range.

Program Continuation: Clear alarm with the RESET key. Restart part program

15800 Channel %1 block %2 wrong starting conditions for CONTPRON/CONTDCON

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: There is an error in the start conditions for CONTPRON/CONDCON:
- G40 not active
- SPLINE or POLY active

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

15810 Channel %1 block %2 wrong array dimension for CONTPRON/CONTDCON

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The number of columns for the array created for CONTPRON/CONTDCON does not conform to the current programming guide.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

15900 Channel %1 block %2 touch probe not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Measure with deletion of distance-to-go
In the part program, an illegal probe has been programmed with the command MEAS (measure with deletion of distance-to-go). The probe numbers
0 ... no probe
1 ... probe 1
2 ... probe 2
are allowed, whether the probe is actually connected or not.
Example:
N10 MEAS=2 G01 X100 Y200 Z300 F1000
Probe 2 with deletion of distance-to-go

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Include a probe number within the limits given above in the keyword MEAS=... This must correspond to the hardware connection of the probe.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15910 Channel %1 block %2 touch probe not allowed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Measure without deletion of distance-to-go

In the part program, an illegal probe has been programmed with the command MEAW (measure without distance-to-go). The probe numbers

0 ... no probe

1 ... probe 1

2 ... probe 2

are allowed, whether the probe is actually connected or not.

Example:

N10 MEAW=2 G01 X100 Y200 Z300 F1000

Probe 2 without deletion of distance-to-go

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Include a probe number within the limits given above in the keyword MEAW=... This must correspond to the hardware connection of the probe.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15950 Channel %1 block %2 no traverse motion programmed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Measure with deletion of distance-to-go

In the part program, no axis or a traversing path of zero has been programmed with the command MEAS (measure with deletion of distance-to-go).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the part program and add the axis address or the traversing path to the measurements block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

15960 Channel %1 block %2 no traverse motion programmed

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: Measure without deletion of distance-to-go

In the part program, no axis or a traversing path of zero has been programmed with the command MEAW (measure without deletion of distance-to-go).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Correct the part program and add the axis address or the traversing path to the measurements block.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16000 Channel %1 block %2 invalid value for lifting direction

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: During the "rapid lift from contour" (keyword: LIFTFAST), a code value for the lifting direction (keyword: ALF=...) which lies outside the permissible range (permissible value range: 0 to 8) was programmed .
With active cutter radius compensation:
Code numbers 2, 3 and 4 cannot be used in G41
Code numbers 6, 7 and 8 cannot be used in G42 because they code the direction to the contour.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Program the lifting direction under ALF=... within the permissible limits.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16005 Channel %1 block %2 invalid value for lifting distance

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Mistake in programming: the value for the lifting path must not be negative.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16010 Channel %1 block %2 machining stop after lift fast

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: LIFTFAST without interrupt routine (Asup) has been programmed. The channel is stopped after the lift motion has been carried out.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: After the channel stop, the axes must be retracted manually in JOG and the program aborted with Reset.
Program Continuation: Clear alarm with the RESET key. Restart part program

16015 Channel %1 block %2 wrong axis identifier %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name
Definitions: Axis names from different coordinate systems were used to program axes for LIFTFAST. The retraction movement is no longer clear.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Use axis names from one coordinate system.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16016 Channel %1 block %2 no retraction position programmed for axis %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name

Definitions: The retraction enable was programmed for LIFTFAST without defining a retraction position for the axis. The retraction movement is no longer clear.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program a retraction position for the relevant axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16020 Channel %1 repositioning in block %2 is not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Programming or operator action incorrect:
Repositioning via REPOS command is only possible in an asynchronous subprogram (interrupt routine).
If the REPOS command was programmed, e.g. in the main program or in a cycle, part program execution is aborted with alarm 16020.
In addition, the alarm is output in the following situations:

- Access to $AC_RETPOINT (repositioning point) outside an ASUP (e.g. in the main program)
- An axis to be repositioned was a oscillating axis with sychronous infeed (OSCILL) in the interrupted block and is now in a state that does not allow it to be traversed as a oscillating axis. Remedy: Change the axis to "neutral axis" state before repositioning with WAITP.
- An axis to be repositioned was an infeed axis for a oscillating axis in the interrupted block; now it can no longer be traversed as one. Remedy: Change the axis back to "POS axis" state before repositioning.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify the part program if necessary.

Program Continuation: Clear alarm with the RESET key. Restart part program

16100 Channel %1 block %2 spindle %3 not available in the channel

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String

Definitions: Mistake in programming: This channel does not recognize the spindle number. The alarm can occur together with a dwell or SPI function.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Check the part program to determine whether the programmed spindle number is correct and whether the program is run in the correct channel.

Check MD 35000 SPIND_ASSIGN_TO MACHAX for all machine axes to see whether one of them contains the programmed spindle number. This machine axis number must be entered in a channel axis of the channel-specific machine data 20070 AXCONF_MACHAX_USED.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16105 Channel %1 block %2 spindle %3 cannot be assigned

Parameters: %1 = Channel number

%2 = Block number, label

%3 = String

Definitions: Mistake in programming: The programmed spindle is not assigned a real axis by the spindle number converter. The alarm can be issued after improper use of $SC_SPIND_ASSIGN_TAB[].

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Correct setting data or modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

16110 Channel %1 block %2 spindle %3 for dwell time not in control mode

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Axis, spindle

Definitions: The spindle can be in the positioning mode, oscillating mode and control mode. With the M command M70 it can be changed from a spindle to an axis. The control mode is divided into the speed-controlled and position-controlled mode, and it is possible to alternate between these with the keywords SPCON and SPCOF.

Positioning mode:

Position control (spindle position under SPOS/SPOSA)

Oscillating mode:

Speed control (M41 - M45 or M40 and S...)

Control mode:

Speed control (spindle speed under S..., M3/M4/M5)

Position control (SPCON/SPCOF, spindle speed under S..., M3/M4/M5)

Axis mode:

Position control (M70/M3, M4, M5, axis position under user-selectable axis name)

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check part program for correct spindle number.

With M3, M4 or M5 put the required spindle into control mode before calling the dwell time.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16120 Channel %1 block %2 invalid index for tool fine compensation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Mistake in programming: The 2nd parameter in the PUTFTOC command indicates for which tool parameter the value is to be corrected (1 - 3 tool lengths, 4 tool radius). The programmed value is beyond the permitted range.
Permissible values are 1 - 4 if on-line tool radius compensation is allowed (see machine data ONLINE_CUTCOM_ENABLE), otherwise values 1 - 3.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program: Length 1 - 3 or 4 permissible for radius.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16130 Channel %1 block %2 instruction not allowed with FTOCON

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:
- Case 1: A plane change is not allowed if the modal G function FTOCON: "fine tool compensation" is active.
- Case 2: Transformation selection is allowed only for zero transformation or transformation inclined axis, Transmit or Tracyl if FTOCON is active.
- Case 3: Tool change is not allowed with M06 if FTOCON has been active since the last tool change.
- Case 4: Orientable tool holder is active.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program: Deselect fine tool compensation with FTOCOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16140 Channel %1 block %2 FTOCON not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool fine compensation (FTOC) is not compatible with the currently active transformation.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program: Deselect fine tool compensation with FTOCOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16150 Channel %1 block %2 invalid spindle number with PUTFTOCF

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The spindle number programmed for PUTFTOC or PUTFTOCF is beyond the permitted range for the spindle numbers.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program. Is the programmed spindle number available?

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16200 Channel %1 block %2 spline and polynominal interpolation not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The spline and polynomial interpolation are options that are not contained in the basic version of the control.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Do not program spline and polynomial interpolation, or retrofit the necessary option.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16300 Channel %1 block %2 denominator polynominal with zero places within parameter range not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed denominator polynomial (with PL [ ] = ... , i.e. without specification of geometry axis) has a zero place within the defined parameter range (PL = ...). This means that the quotient of the numerator polynomial and the denominator polynomial is infinite or indeterminate.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify the polynomial block so that there is no zero place within the polynomial length in the denominator polynomial.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16400 Channel %1 block %2 positioning axis %3 cannot participate in spline

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: An axis assigned to a spline grouping (n) with SPLINEPATH (n, AX1, AX2, ...) has been programmed as positioning axis with POS or POSA.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Do not assign the positioning axis to the spline grouping.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16410 Channel %1 block %2 axis %3 is not a geometry axis

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: A geometry axis has been programmed that cannot be imaged on any machine axis in the current transformation (possibly there is no transformation active at the moment).
Example:
Without transformation: Polar coordinate system with X, Z, and C axis
With transformation: Cartesian coordinate system with X, Y, and Z, e.g. with TRANSMIT.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Activate transformation type with TRAORI (n) or do not program geometry axes that do not participate in the transformation grouping.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16420 Channel %1 block %2 axis %3 programmed repeatedly

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: It is not allowed to program an axis more than once.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Delete the axis addresses that have been programmed more than once.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16421 Channel %1 block %2 angle %3 programmed repeatedly in the block

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Angle

Definitions: It is not allowed to program more than one PHI or PSI angle for an orientation vector in the same block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16422 Channel %1 block %2 angle %3 programmed repeatedly in the block

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Angle

Definitions: It is not allowed to program more than one rotation angle THETA for the orientation in one block. The angle of rotation can either be programmed explicitly with THETA or by programming with Euler angles or RPY angles.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16423 Channel %1 block %2 angle %3 programmed repeatedly in the block

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Angle

Definitions: It is not allowed to program more than one polynomial for the orientation rotation angle with PO[THT] in one block.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16424 Channel %1 block %2 coordinate %3 programmed repeatedly in the block

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Coordinate

Definitions: It is not allowed to program a coordinate of the 2nd contact point of the tool for description of the tool orientation several times in one block.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16430 Channel %1 block %2 geometry axis %3 cannot traverse as positioning axis in rotated coordinate system

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: In the rotated coordinate system, traversing of a geometry axis as positioning axis (i.e. along its axis vector in the rotated coordinate system) would mean traversing of several machine axes. This is in conflict with the positioning axis concept, however, in which one axis interpolator runs in addition to the path interpolator!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Traverse geometry axes as positioning axes only with rotation deactivated.
Deactivate rotation:
Keyword ROT without further specification of axis and angle.
Example: N100 ROT

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16440 Channel %1 block %2 rotation programmed for non-existent geometry axis

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A rotation of a geometry axis which does not exist was programmed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16500 Channel %1 block %2 chamfer or rounding negative

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: A negative chamfer or rounding has been programmed under the keywords CHF= ..., RND=... or RNDM=...

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Values for chamfers, roundings and modal roundings must be programmed with positive values only.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16510 Channel %1 block %2 no facing axis available

Parameters:
%1 = Channel number
%2 = Block number, label

Definitions: Diameter programming has been activated with the keyword DIAMON although no facing axis has been programmed in this NC block.
If the diameter axis is not a geometry axis, in the initial setting "DIAMON" the alarm appears as soon as the control is switched on.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Activate the modal G function DIAMON only in NC blocks containing a facing axis or deactivate diameter program with DIAMOF.
In machine data 20150 GCODE_RESET_VALUES[28] select "DIAMOF" for the initial setting.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16700 Channel %1 block %2 axis %3 invalid feed type

Parameters:
%1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: In a thread cutting function, the feed has been programmed in a unit that is impermissible.
1. G33 (thread with constant lead) and the feed have not been programmed with G94 or G95.
2. G33 (thread with constant lead) is active (modal) and G63 is programmed additionally in a following block .conflict situation! (G63 is in the 2nd G group, G33, G331 and G332 are in the 1st G group).
3. G331 or G332 (rigid tapping) and the feed have not been programmed with G94.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Use only the feed type G94 or G95 in the thread cutting functions.
After G33 and before G63, deselect the thread cutting function with G01.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16710 Channel %1 block %2 axis %3 master spindle not programmed

Parameters:
%1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: A master spindle function has been programmed (G33, G331, G95, G96) but the speed or the direction of rotation of the master spindle is missing.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Add S value or direction of rotation for the master spindle in the displayed block.

Program Continuation: Clear alarm with the RESET key. Restart part program

16715 Channel %1 block %2 axis %3 spindle not in standstill

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Spindle number

Definitions: In the applied function (G74, reference point approach), the spindle must be stationary.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program M5 or SPOS/SPOSA in front of the defective block in the part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16720 Channel %1 block %2 axis %3 thread lead is zero

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: No lead was programmed in a thread block with G33 (thread with constant lead) or G331 (rigid tapping).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: The thread lead must be programmed for the specified geometry axis under the associated interpolation parameters.
X -> I
Y -> J
Z -> K

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16730 Channel %1 block %2 axis %3 wrong parameter

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: In G33 (tapping with constant lead) the lead parameter was not assigned to the axis that determines the velocity.
For longitudinal and face threads, the thread lead for the specified geometry axis must be programmed under the associated interpolation parameter.
X -> I
Y -> J
Z -> K
For taper threads, the address I, J, K depends on the axis with the longer path (thread length). A 2nd lead for the other axis is, however, not specified.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Assign lead parameters to the axis that determines the velocity.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16740 Channel %1 block %2 no geometry axis programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No geometry axis was programmed for tapping (G33) or for rigid tapping (G331, G332). The geometry axis is, however, essential if an interpolation parameter has been specified.
Example:
N100 G33 Z400 K2 ; thread lead 2mm, thread end Z=400 mm
N200 SPOS=0 ; position spindle in axis mode
N201 G90 G331 Z-50 K-2 ; tapping to Z=-50, counterclockwise
N202 G332 Z5 ; retraction, direction reversal automatic
N203 S500 M03 ; spindle again in spindle mode
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Specify geometry axis and corresponding interpolation parameters.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16750 Channel %1 block %2 axis %3 SPCON not programmed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: For the programmed function (rotary axis, positioning axis), the spindle must be in position control mode.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Program position control of the spindle with SPCON in the previous block.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16751 Channel %1 block %2 spindle/axis %3 SPCOF not executable

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: For the programmed function, the spindle must be in the open-loop control mode. In the positioning or axis mode, the position control must not be deselected.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Put the spindle into open-loop control mode in the preceding block. This can be done with M3, M4 or M5 for the relevant spindle.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16755 Channel %1 block %2 no stop required

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: No Stop is needed for the programmed function. A Stop is necessary after SPOSA or after M5 if the next block is to be applied only after the spindle has come to a stop.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Do not write instruction.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16760 Channel %1 block %2 axis %3 S value missing

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: No spindle speed has been given for rigid tapping (G331 or G332).
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Program the spindle speed under address S in [rpm] (in spite of axis mode); the direction of rotation is given by the sign of the spindle lead.
Positive thread lead: Rotational direction as M03.
Negative thread lead: Rotational direction as M04.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16761 Channel %1 block %2 axis/spindle %3 not programmable in the channel

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: Mistake in programming: The axis/spindle cannot be programmed in the channel at this time. This alarm can occur when the axis/spindle is being used by another channel or by the PLC.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program, use "GET()".
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16762 Channel %1 block %2 spindle %3 thread function is active

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Spindle number
Definitions: Mistake in programming: The spindle function cannot be executed at the present time. This alarm occurs when the spindle (master spindle) is linked with the axes by an interpolation function.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program. Deselect thread cutting or tapping.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16763 Channel %1 block %2 axis %3 programmed speed is illegal (zero or negative)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: A spindle speed (S value) was programmed with the value zero or with a negative value.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: The programmed spindle speed (S value) must be positive. Depending on the application case, the value zero can be accepted (e.g. G25 S0).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16770 Channel %1 block %2 axis %3 no measuring system available

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: SPCON, SPOS or SPOSA has been programmed. These functions require at least one measuring system. According to MD: NUM_ENCS the machine axis/spindle has no measuring system.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Please inform the authorized personnel/service department. Retrofit a measuring system.

Program Continuation: Clear alarm with the RESET key. Restart part program

16771 Channel %1 following axis %2 overlaid movement not enabled

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: No gear synchronization and no overlay movement can be executed because this is not enabled at the VDI interface.

Reactions: - Alarm display.

Remedy: Set the "enable following axis overlay" VDI signal.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

16776 Channel %1 block %2 curve table %3 does not exist for axis %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table
%4 = Axis name, spindle number

Definitions: An attempt was made to couple axis %4 with curve table number %3, but no curve table of this number exists.

Reactions:
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Modify the NC part program so that the required curve table exists when axis link is to be activated.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

16777 Channel %1 block %2 coupling: following axis %3 for lead axis %4 not available

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: A coupling has been switched on in which the slave spindle/axis is currently not available. Possible causes:
- The spindle/axis is active in the other channel.
- The spindle/axis has been accessed by the PLC and has not yet been released.

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Put the master spindle/axis with spindle/axis exchange into the necessary channel or release from the PLC.

Program Continuation: Clear alarm with the RESET key. Restart part program

16778 Channel %1 block %2 coupling: Ring coupling at following axis %3 and leading axis %4 impermissible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: A coupling has been switched on which results in a cyclic coupling, allowance being made for further couplings. This cyclic coupling cannot be uniquely computed.

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Configure link in accordance with the MD or correct NC part program (channel MD: COUPLE_AXIS_n).

Program Continuation: Clear alarm with the RESET key. Restart part program

16779 Channel %1 block %2 coupling: too many couplings for axis %3, see active leading axis %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: More leading axes and spindles were defined for the specified axis/spindle than are allowed. The last parameter to be specified is a leading value object/leading axis to which the specified axis/spindle is already linked.

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

16780 Channel %1 block %2 following spindle/axis missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The following spindle/axis has not been written in the part program.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16781 Channel %1 block %2 master spindle/axis missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The master spindle/axis has not been programmed in the part program.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16782 Channel %1 block %2 following spindle/axis %3 not available

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: A coupling has been switched on in which the slave spindle/axis is currently not available. Possible causes:
- The spindle/axis is active in the other channel.
- The spindle/axis has been accessed by the PLC and has not yet been released.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Put the master spindle/axis with spindle/axis exchange into the necessary channel or release from the PLC.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16783 Channel %1 block %2 master spindle/axis %3 not available

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: A coupling has been switched on in which the master spindle/axis is currently not available. Possible causes:
- Setpoint linkage has been selected and spindle/axis is active in the other channel.
- The spindle/axis has been accessed by the PLC and has not yet been released.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Put the master spindle/axis with spindle/axis exchange into the necessary channel or release from the PLC.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16785 Channel %1 block %2 identical spindles/axes %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: A coupling has been switched on in which the following spindle/axis is identical to the master spindle/axis.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department.
- Configure link accordingly in MD (channel MD: COUPLE_AXIS_n)
- or modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16787 Channel %1 block %2 coupling parameter not changeable

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The specified coupling is write-protected. Therefore, the coupling parameters cannot be modified.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Remove write protection. Channel MD: COUPLE_AXIS_IS_WRITE_PROT
- or modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

16788 Channel %1 block %2 cyclic coupling

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A coupling has been switched on which results in a cyclic coupling, allowance being made for further couplings. This cyclic coupling cannot be uniquely computed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department.
- Configure link accordingly in MD (channel MD: 21300 COUPLE_AXIS_n)
- or modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16789 Channel %1 block %2 multiple link

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A coupling has been switched on in which the axes/spindles have already been assigned by another coupling. Parallel couplings cannot be processed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Check in the part program whether another link already exists for the axes.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16790 Channel %1 block %2 Parameter is zero or missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A coupling has been switched on in which a relevant parameter has been specified with zero or has not been written (e.g. denominator in the transmission ratio, no slave axis).
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department.
- Configure link accordingly in MD (channel MD: 42300 COUPLE_AXIS_n)
- or modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16791 Channel %1 block %2 parameter is not relevant

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A coupling has been switched on in which a non-relevant parameter has been written (e.g. parameter for ELG).
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16792 Channel %1 block %2 too many couplings for axis/spindle %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: For the specified axis/spindle, more master axes/spindles have been defined than are allowed.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16793 Channel %1 block %2 coupling of axis %3 prohibits transformation change

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: The specified axis is a slave axis in a transformation grouping. When the coupling is switched on, the transformation cannot be changed to another one.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program. Switch off coupling(s) of this axis before changing transformation or do not change the transformation.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16794 Channel %1 block %2 coupling of axis/spindle %3 prohibits reference point approach

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: The specified axis is a (gantry) slave axis and cannot therefore approach the reference point.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program. Switch off coupling(s) of this axis before reference point approach or do not reference. A gantry slave axis cannot reference for itself.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16795 Channel %1 block %2 string cannot be interpreted

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A coupling has been switched on in which a non-interpretable string has been written (e.g. block change behavior).
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16796 Channel %1 block %2 coupling not defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A coupling is to be switched the parameters of which have neither been programmed nor configured.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Correct NC part program or MD, program the coupling with COUPDEF or configure by means of MD.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16797 Channel %1 block %2 coupling is active

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An operation is to be performed in which no coupling may be active, e.g. COUPDEL or TANGDEL must not be used on active couplings.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Correct NC part program, deselect the link with COUPOF or TANGOF.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16798 Channel %1 block %2 axis %3 is following axis and prohibits axis container rotation

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: The programmed axis/spindle is active as a slave axis/spindle in a coupling. When the coupling is active, the axis container cannot be rotated.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program. Deactivate the coupling(s) for this axis/spindle before rotating the axis container or execute the axis container rotation at a later time.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16799 Channel %1 block %2 axis %3 is master axis and prohibits axis container rotation

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: The programmed axis/spindle is active as a master axis/spindle in a coupling. When the coupling is active, the axis container cannot be rotated.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Modify part program. Deactivate the coupling(s) for this axis/spindle before rotating the axis container or execute the axis container rotation at a later time.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16800 Channel %1 block %2 traverse instruction DC/CDC for axis %3 not allowed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
Definitions: The keyword DC (Direct Coordinate) can only be used for rotary axes. This causes approach of the programmed absolute position along the shortest path.
Example:
N100 C=DC(315)
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Replace the keyword DC in the displayed NC block by specifying AC (Absolute Coordinate).
If the alarm display is the result of an error in the axis definition, the axis can be declared as a rotary axis by means of the axis-specific MD 30300 IS_ROT_AX.
Corresponding machine data:
Modify MD 30310: ROT_IS_MODULO
Modify MD 30320: DISPLAY_IS_MODULO
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16810 Channel %1 block %2 traverse instruction ACP for axis %3 not allowed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The keyword ACP (Absolute Coordinate Positive) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. In the displayed NC block, replace the keyword ACP by specifying AC (Absolute Coordinate).
If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD 30300: IS_ROT_AX and MD 30310: ROT_IS_MODULO can be declared a rotary axis with modulo change.
Corresponding machine data:
Modify MD 30,320: DISPLAY_IS_MODULO

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16820 Channel %1 block %2 traverse instruction ACN for axis %3 not allowed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The keyword ACN (Absolute Coordinate Negative) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. In the displayed NC block, replace the keyword ACN by specifying AC (Absolute Coordinate).
If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD 30300: IS_ROT_AX and MD 30310: ROT_IS_MODULO can be declared a rotary axis with modulo change.
Corresponding machine data:
Modify MD 30320: DISPLAY_IS_MODULO

Program Continuation: Clear alarm with the RESET key. Restart part program

16830 Channel %1 block %2 incorrect position programmed for axis/spindle %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: A position beyond the range of 0 - 359.999 has been programmed for a modulo axis.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program position in the range 0 - 359.999.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16903 Channel %1 program control: action %2 not allowed in the current state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The relevant action cannot be processed now. This can occur, for instance, during read-in of machine data.
Reactions: - Alarm display.
Remedy: Wait until the procedure is terminated or abort with Reset and repeat the operation.
Program Continuation: Clear alarm with the Delete key or NC START.

16904 Channel %1 program control: action %2 not allowed in the current state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The operation (program, JOG, block search, reference point, etc.) cannot be started or continued in the current status.
Reactions: - Alarm display.
Remedy: Check the program status and channel status.
Program Continuation: Clear alarm with the Delete key or NC START.

16905 Channel %1 program control: action %2 not allowed

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: Operation cannot be started or continued. A start is only accepted when an NCK function can be started.
Example: A start is accepted in JOG mode when, for example, the function generator is active or a JOG movement has first been stopped with the Stop key.
Reactions: - Alarm reaction in Automatic mode.
Remedy: Check the program status and channel status.
Program Continuation: Clear alarm with the Delete key or NC START.

16906 Channel %1 program control: action %2 is aborted due to an alarm

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The action was aborted due to an alarm.
Reactions: - Alarm display.
Remedy: Eliminate the error and acknowledge the alarm. Then repeat the operation.
Program Continuation: Clear alarm with the Delete key or NC START.

16907 Channel %1 action %2 only possible in stop state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: This action may only be performed in Stop state.
Reactions: - Alarm display.
Remedy: Check the program status and channel status.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

16908 Channel %1 action %2 only possible in reset state or at the block end

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: This action may only be performed in Reset state or at end of block.
Reactions: - Alarm display.
Remedy: Check the program status and channel status.
Program Continuation: Clear alarm with the Delete key or NC START.

16909 Channel %1 action %2 not allowed in current mode

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: You have to activate a different operating mode for the function to be activated.
Reactions: - Alarm display.
Remedy: Check operation and operating state.
Program Continuation: Clear alarm with the Delete key or NC START.

16911 Channel %1 mode change is not allowed

Parameters: %1 = Channel number
Definitions: The change from overstoring into another operating mode is not allowed.
Reactions: - Alarm display.
Remedy: After overstoring is terminated, it is possible to change to another operating state again.
Program Continuation: Clear alarm with the Delete key or NC START.

16912 Channel %1 program control: action %2 only possible in reset state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: This action can only be performed in Reset state.
Example: Program selection through MMC or channel communication (INIT) can only be performed in Reset state.
Reactions: - Alarm display.
Remedy: Reset or wait until processing is terminated.
Program Continuation: Clear alarm with the Delete key or NC START.

16913 Mode group %1 channel %2 mode change: action %3 not allowed

Parameters: %1 = Channel number
%2 = Mode group number
%3 = Action number/action name
Definitions: The change to the desired mode is not permitted. The change can only take place in the Reset state.
Example: Program processing is halted in AUTO mode by NC Stop. Then there is a mode change to JOG mode (program status interrupted). From this operating mode it is only possible to change to AUTO mode and not to MDA mode!
Reactions: - Alarm display.
Remedy: Either activate the Reset key to reset program processing, or activate the mode in which the program was being processed previously.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

16914 Mode group %1 channel %2 mode change: action %3 not allowed

Parameters: %1 = Channel number
%2 = Mode group number
%3 = Action number/action name
Definitions: Incorrect mode change, e.g.: Auto -> MDAREF.
Reactions: - Alarm display.
Remedy: Check operation or selected mode.
Program Continuation: Clear alarm with the Delete key or NC START.

16915 Channel %1 action %2 not allowed in the current block

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: If traversing blocks are interrupted by asynchronous subroutines, then it must be possible for the interrupted program to continue (reorganization of block processing) after termination of the asynchronous subroutine.
The 2nd parameter describes which action wanted to interrupt block processing.
Reactions: - Alarm display.
Remedy: Let the program continue to a reorganized NC block or modify part program.
Program Continuation: Clear alarm with the Delete key or NC START.

16916 Channel %1 repositioning: action %2 not allowed in the current state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: Repositioning of block processing presently not possible. In certain cases this can prevent a mode change from taking place.
The 2nd parameter describes which action should be used to perform repositioning.
Reactions: - Alarm display.
Remedy: Let the program continue to a repositioned NC block or modify part program.
Program Continuation: Clear alarm with the Delete key or NC START.

16918 Channel %1 for action %2 all channels must be in reset state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: All channels must be in the initial setting in order to carry out the action! (For example, for machine data loading)
Reactions: - Alarm display.
Remedy: Either wait until the channel status is aborted or press the Reset key.
Program Continuation: Clear alarm with the Delete key or NC START.

16919 Channel %1 action %2 is not allowed due to a pending alarm

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: This action cannot be performed due to an alarm, or the channel is in fail.
Reactions: - Alarm display.
Remedy: Press RESET key.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

16920 Channel %1 action %2 is already active

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: An identical action is still active.
Reactions: - Alarm display.
Remedy: Wait until the first procedure is terminated or abort with Reset and repeat the operation.
Program Continuation: Clear alarm with the Delete key or NC START.

16921 Channel %1 mode group %2 machine data: channel/mode group assignment not allowed or assigned twice

Parameters: %1 = Channel number
%2 = Mode group number
Definitions: On powering up, an illegal channel/mode group assignment was detected.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.
- Mode group not ready.
- Channel not ready.

Remedy: Please inform the authorized personnel/service department. Check machine data ASSIGN_CHAN_TO_MODE_GROUP.
Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

16922 Channel %1 subprograms: action %2 maximum nesting depth exceeded

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines are activated. These asynchronous subroutines can be interrupted in the same manner as user programs. Unlimited nesting depth is not possible for asynchronous subroutines due to memory limitations.
Example: An interrupt interrupts the current program processing. Other interrupts with higher priorities interrupt processing of the previously activated asynchronous subroutines.
Possible actions are: DryRunOn/Off, DecodeSingleBlockOn, delete distance-to-go, interrupts .....
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Do not trigger the event on this block.
Program Continuation: Clear alarm with the RESET key. Restart part program

16923 Channel %1 program control: action %2 not allowed in the current state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The current processing cannot be stopped, due to an active preprocessing process. This applies to, for example, loading machine data and block searches until the search object is found.
Reactions: - Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Abort by pressing Reset!
Program Continuation: Clear alarm with the Delete key or NC START.

16924 Channel %1 caution: program test modifies tool management data

Parameters: %1 = Channel number
Definitions: Tool management data is changed during program testing. It is not possible to automatically rectify the data after termination of the program testing.
This error message prompts the user to make a backup copy of the data or to reimport the data after the operation is terminated.
Reactions: - Alarm display.
Remedy: Please inform the authorized personnel/service department. Save tool data on MMC and reimport data after "ProgtestOff".
Program Continuation: Clear alarm with the Delete key or NC START.

16925 Channel %1 program control: action %2 not allowed in the current state, action %3 active

Parameters: %1 = Channel number
%2 = Action number/action name
%3 = Action number/action name
Definitions: The action has been refused since a mode or sub-mode change (change to automatic mode, MDA, JOG, overstoring, digitizing, etc.) is taking place.
Example: This alarm message is output if the Start key is pressed during a mode or sub-mode change from, for example, automatic to MDA, before the NCK has confirmed selection of the mode.
Reactions: - Alarm display.
Remedy: Repeat action.
Program Continuation: Clear alarm with the Delete key or NC START.

16926 Channel %1 channel coordination: action %2 not allowed in block %3, marker %4 is already set

Parameters: %1 = Channel number
%2 = Action
%3 = Block number
%4 = Marker number
Definitions: The action was denied, the marker was already set. Check the program.
Example:
SETM(1) ; CLEARM(1) ; Marker must be reset first.
SETM(1)
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Repeat action.
Program Continuation: Clear alarm with the RESET key. Restart part program

16927 Channel %1 action %2 at active interrupt treatment not allowed

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: This action may not be activated during interrupt processing (e.g. mode change).

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
Remedy: Reset or wait until interrupt processing is terminated.
Program Continuation: Clear alarm with the Delete key or NC START.

16928 Channel %1 interrupt treatment: action %2 not possible

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: A program interrupt has been activated in a non REORG capable block.
Examples of possible program interrupt in this case:
- Traversing to fixed stop
- VDI channel delete distance-to-go
- VDI axial delete distance-to-go
- Measuring
- Software limit
- Axis replacement
- Axis from correction
- Servo disable
- Gear stage change at actual gear stage unequal to setpoint gear stage.

The relevant block concerns a:
- Pick-up block during block search (excluding last pick-up block)
- Block in overstore interrupt.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Do not trigger the event on this block.
Program Continuation: Clear alarm with the RESET key. Restart part program

16930 Channel %1: preceding block and current block %2 must be separated through an executable block

Parameters: %1 = Channel number
%2 = Block number
Definitions: The language functions WAITMC, SETM, CLEARM and MSG must be packed in separate NC blocks due to the language definition. To avoid velocity drops, these blocks are attached to the next NC block internally in the NCK (for MSG only in path control mode, for WAITMC to the previous NC block). For this reason, there must always be an executable block (not a calculation block) between the NC blocks. An executable NC block always includes e.g. travel movements, a help function, Stopre, dwell time etc.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- Correction block is reorganized.

Remedy: Program an executable NC block between the previous and the current NC block.
Example:
N10 SETM.
N15 STOPRE ; insert executable NC block.
N20 CLEARM.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

16931 Channel %1 subprograms: action %2 maximum nesting depth exceeded

Parameters: %1 = Channel number

%2 = Action number/action name

Definitions: Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines are activated. These asynchronous subroutines can be interrupted in the same manner as the user program. Unlimited nesting depth is not possible for asynchronous subroutines due to memory limitations.

Example: In the case of an approach block in a repositioning procedure do not interrupt repeatedly, instead wait until processing is completed.

Possible actions are: mode change, SlashOn/Off, overstoring.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.

Remedy: Initiate a block change and repeat the action.

Program Continuation: Clear alarm with the Delete key or NC START.

16932 Channel %1 conflict when activating user data type %2

Parameters: %1 = Channel number

%2 = Data type

Definitions: The "activate user data" function (PI service _N_SETUDT) modifies a data block (tool offset, settable zero offset or base frame) which is also written by the NC blocks in preparation.

In the event of a conflict, the value entered by the MMC is reset.

Parameter %2 specifies which data block is affected:

1: Active tool offset

2: Base frame

3: Active zero offset

Reactions: - Alarm display.

Remedy: Check the inputs on the MMC and repeat if necessary.

Program Continuation: Clear alarm with the Delete key or NC START.

16933 Channel %1 interrupt treatment: action %2<ALNX> not allowed in the current state

Parameters: %1 = Channel ID

%2 = Action number/action name

Definitions: If a temporary standstill has occurred because of a Reorg event across block boundaries, it is possible that a block without Reorg capability has been loaded. In this situation, it is unfortunately necessary to abort the Reorg event handling! Reorg events are, e.g. abort subprogram, delete distance-to-go and interrupts.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Abort program with the RESET key.

Program Continuation: Clear alarm with the RESET key. Restart part program

16934 Channel %1 interrupt treatment: action %2<ALNX> not possible due to stop

Parameters: %1 = Channel ID

%2 = Action number/action name

© Siemens AG, 2002. All rights reserved

Definitions: Reorg events are, e.g. abort subprogram, delete distance to go and interrupts, axis replacement, termination of follow-up mode. Two Reorg events overlap in this situation. The 2nd Reorg event coincides with the 1st block generated by the previous event. (e.g. an axis replacement is induced twice in rapid succession). Axis replacement leads to Reorg in the channels in which an axis is removed without preparation. This block must be stopped in the above sequence in order to prevent the interpolator buffer from overflowing. This can be achieved by pressing the Stop or StopAll key, configuring an alarm with INTERPRETERSTOP or by decode single block.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: The program must be aborted with Reset.

Program Continuation: Clear alarm with the RESET key. Restart part program

16935 Channel %1 action %2<ALNX> not possible due to search run

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: The action is not allowed as block search is currently running via program test. Block search via program test: "PI Service _N_FINDBL with mode parameter 5”.
With this block search type, it is not permissible to activate program test or dry run feedrate.

Reactions: - Alarm display.

Remedy: Activate the action after block search is terminated.

Program Continuation: Clear alarm with the Delete key or NC START.

16936 Channel %1 action %2<ALNX> not possible due to active dry run

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: This action is not allowed as dry run feedrate is currently active.
Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when dry run feedrate is active.

Reactions: - Alarm display.

Remedy: Abort program with the RESET key.

Program Continuation: Clear alarm with the Delete key or NC START.

16937 Channel %1 action %2<ALNX> not possible due to program test

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: This action is not allowed as program test is currently active.
Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when program test is active.

Reactions: - Alarm display.

Remedy: Deactivate program test.

Program Continuation: Clear alarm with the Delete key or NC START.

16938 Channel %1 action %2<ALNX> aborted due to active gear change

Parameters: %1 = Channel ID
%2 = Action number/action name

© Siemens AG, 2002. All rights reserved

Definitions: Reorganization events are, among others, subprogram abort, delete distance-to-go and interrupts, axis replacement, exiting the correction state. These events wait for the end of a gear change. However, the maximum waiting period has elapsed.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Program must be aborted with Reset and, if necessary, GEAR_CHANGE_WAIT_TIME must be increased.

Program Continuation: Clear alarm with the RESET key. Restart part program

16939 Channel %1 action %2 rejected due to active gear change

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: Reorganization events that are possible in Stop state, e.g mode change, are waiting for the end of the gear change. However, the maximum waiting period has elapsed.

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Repeat action or increase MD GEAR_CHANGE_WAIT_TIME.

Program Continuation: Clear alarm with the Delete key or NC START.

16940 Channel %1 action %2<ALNX> wait for gear change

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: Reorganization events are waiting for the end of a gear change. The alarm is displayed during the waiting period.

Reactions:
- Alarm display.
- Warning display.

Remedy: Alarm is suppressed by means of ENABLE_ALARM_MASK bit 1 == 0.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

16941 Channel %1 action %2<ALNX> rejected because no program event has been executed yet

Parameters: %1 = Channel ID
%2 = Action number/action name

Definitions: The setting of the machine data $MC_PROG_EVENT_MASK forces an asynchronous subprogram to be triggered automatically on RESET or PowerOn. The implicitly triggered asynchronous subprograms are normally called "Event-triggered program call" or "Program event".

In the alarm situation, this asynchronous subprogram could not yet be activated; that is why the action (normally start of part program) must be rejected.

Reasons for the fact that the asynchronous subprogram could not be triggered:

1. The asynchronous subprogram does not exist (/_N_CMA_DIR/ _N_PROG_EVENT_SPF)
2. The asynchronous subprogram is allowed to start in the referenced state only (see $MN_ASUP_START_MASK)
3. READY is missing (because of alarm)

Reactions:
- Alarm display.

Remedy:
- Load program
- Check $MN_ASUP_START_MASK
- Acknowledge alarm

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the Delete key or NC START.

16942 Channel %1 start program command action %2<ALNX> not possible

Parameters: %1 = Channel ID

%2 = Action number/action name

Definitions: Currently, the alarm occurs only in combination with the SERUPRO action. SERUPRO stands for search via program test.

SERUPRO is currently searching the search target and has therefore switched this channel to the program test mode. With the START program command in channel 1, another channel 2 would actually be started, which means that axes would really be started during the search action.

If this alarm is switched off (see help), the user can make use of the above behavior by initially selecting via PLC the program test mode in channel 2, leaving channel 2 executing until its natural end, stopping channel 2 in order to deselect program test again.

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Alarm can be switched off with $MN_SERUPRO_MASK bit 1.

Program Continuation: Clear alarm with the RESET key. Restart part program

16943 Channel %1 action %2<ALNX> not possible due to ASUP

Parameters: %1 = Channel ID

%2 = Action number/action name

Definitions: The action in the 2nd parameter was rejected, since an asynchronous subprogram is currently active.

Currently, only the integrated search run is rejected with this alarm. The integrated search run is activated, if search run is triggered in the Stop program state. In other words: Parts of a program have already been executed and a following program part is "skipped" with search run in order to continue the program afterwards.

The event is not possible if the program is stopped within an asynchronous subprogram or if an asynchronous subprogram had been selected before the event. An asynchronous subprogram is selected, when the triggering asynchronous subprogram event arrives, but the asynchronous subprogram cannot be started (e.g. the asynchronous start program is not started because of a read-in disable or because the Stop key is active).

In this case, it is irrelevant whether a user ASUP or a system ASUP has been triggered. User ASUPs are activated via FC-9 or via the fast inputs.

The following events lead to system ASUPS:

- Mode change
- Overstore on
- Aborting subprogram level
- Switching on of single block, type 2
- Setting machine data effective
- Setting user data effective
- Change skip levels
- Dry run on/off
- Program test off
- Correction block alarms
- Editing modi in Teach
- External zero offset

© Siemens AG, 2002. All rights reserved

- Axis replacement
- Delete distance-to-go
- Measuring

Reactions: - Alarm display.
Remedy: Repeat the action after the end of the asynchronous subprogram.
Program Continuation: Clear alarm with the Delete key or NC START.

16944 Channel %1 action %2<ALNX> not possible due to active search blocks

Parameters: %1 = Channel ID
%2 = Action number/action name
Definitions: The NCK is currently processing either the action blocks of the search run or the approach motion after the search run.
In this situation, the action (2nd parameter of the alarm) must be rejected.
Currently, only the integrated search run is rejected with this alarm. The integrated search run is activated, if search run is triggered in the Stop program state. In other words: Parts of a program have already been executed and a following program part is "skipped" with search run in order to continue the program afterwards.
Reactions: - Alarm display.
Remedy: Repeat the action after the approach motion of the search run.
Program Continuation: Clear alarm with the Delete key or NC START.

16945 Channel %1 action %2<ALNX> delayed up to the block end

Parameters: %1 = Channel ID
%2 = Action number/action name
Definitions: The currently executing action (e.g. dry run on/off, change skip levels, etc.) should be active immediately, but it can become active not earlier than at the end of the block, since a thread is currently being machined. The action is activated with a slight delay.
Example: Dry run is started in the middle of the thread, then traversing at high speed does not start before the next block.
Reactions: - Alarm display.
Remedy: Alarm can be switched off via $MN_SUPPRESS_ALARM_MASK bit 17==1.
Program Continuation: Clear alarm with the Delete key or NC START.

16946 Channel %1 start via START is not allowed

Parameters: %1 = Channel ID
Definitions: This alarm is active with "Group Serupro" only. “Group Serupro" is activated by means of "$MC_SERUPRO_MODE BIT2" and enables the retrace support of entire channel groups during block search.
The machine data $MC_DISABLE_PLC_START specifies which channel is generally started from the PLC and which channel is only allowed to be started from another channel via the START part program command.
This alarm occurs if the channel was started via the START part program command and $MC_DISABLE_PLC_START==FALSE was set.
Reactions: - Alarm display.
Remedy: Modify $MC_DISABLE_PLC_START of switch off "Group Serupro" (see $MC_SERUPRO_MODE).
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

16947 Channel %1 start via PLC is not allowed

Parameters: %1 = Channel ID

Definitions: This alarm is active with "Group Serupro" only. "Group Serupro" is activated by means of "$MC_SERUPRO_MODE BIT2" and enables the retrace support of entire channel groups during block search.

The machine data $MC_DISABLE_PLC_START specifies which channel is generally started from the PLC and which channel is only allowed to be started from another channel via the START part program command.

This alarm occurs if the channel was started via the PLC and $MC_DISABLE_PLC_START==TRUE was set.

Reactions: - Alarm display.

Remedy: Modify $MC_DISABLE_PLC_START of switch off "Group Serupro" (see $MC_SERUPRO_MODE).

Program Continuation: Clear alarm with the Delete key or NC START.

16948 Channel %1 dependent channel %2 still active

Parameters: %1 = Channel ID

%2 = Channel ID

Definitions: This alarm is active with "Group Serupro" only. “Group Serupro" is activated by means of "$MC_SERUPRO_MODE BIT2" and enables the retrace support of entire channel groups during block search.

A “dependent channel” is a channel that had indirectly been started by the currently active channel. The currently active channel was started via PLC.

This channel m_u_s_t be terminated (i.e. reached M30) before the current channel is terminated.

This alarm occurs if the currently active channel is terminated before the dependent channel.

Reactions: - Alarm display.

Remedy: Switch off "Group Serupro" (see $MC_SERUPRO_MODE) or install WAITE.

Program Continuation: Clear alarm with the Delete key or NC START.

16949 Correspondence between marker of channel %1 and channel %2 is invalid.

Parameters: %1 = Channel ID

%2 = Channel ID

Definitions: This channel defines a WAIT marker with other channels, which on their part have no correspondence with this wait marker.

This channel's WAIT marker has no explicit counterpart in the other channel; i.e. the channels do not mutually wait.

==========

Example

Ch 3 Ch 5 Ch 7

WAITM(99,3,5) WAITM(99,3,5) WAITM(99,5,7)

The wait markers in channels 3 and 5 mutually wait for each other and channel 7 only waits for channel 5. Therefore, channel 7 may continue when 5 and 7 have reached the wait marker, but channel 3 is still far in front of the wait marker.

When it continues, channel 7 deletes its wait marker. When wait marker 99 is reached again, you can no longer determine the behavior precisely.

==========

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: In each wait marker, list all channels with which you want to synchronize, or suppress the alarm with $MN_SUPPRESS_ALARM_MASK, bit 23.

==========

Sample solution A:

Ch 3 Ch 5 Ch 7

WAITM(99,3,5,7) WAITM(99,3,5,7) WAITM(99,3,5,7)

==========

Sample solution B:

Ch 3 Ch 5 Ch 7

WAITM(99,3,5) WAITM(99,3,5)

WAITM(88.50.7) WAITM(88.50.7)

==========

Sample solution C:

Ch 3 Ch 5 Ch 7

WAITM(88.50.7) WAITM(88.50.7)

WAITM(99,3,5) WAITM(99,3,5)

Program Continuation: Clear alarm with the Delete key or NC START.

16950 Channel %1 search run with hold block

Parameters: %1 = Channel ID

Definitions: Informational alarm.

The search run was not performed on the interruption block, instead, it touches down shortly before that. This so-called "hold block" is generated by the part program command IPTRLOCK, or implicitly defined by $MC_AUTO_IPTR_LOCK. This is to preven you from performing a search run in critical program areast (e.g. gear hobbing).

The alarm also displays that, instead of searching for the block that actually was interrupted before, another block is being searched for.
This behavior is desired and the alarm serves only informational purposes.

Reactions: - Alarm display.

Remedy: $MN_SUPPRESS_ALARM_MASK $MC_AUTO_IPTR_LOCK and language command IPTRLOCK

Program Continuation: Clear alarm with the Delete key or NC START.

16951 Channel %1 search run in a program section that cannot be searched

Parameters: %1 = Channel ID

Definitions: With the language commands IPTRLOCK and IPTRUNLOCK, the part programmer can identify part program sections that cannot be searched. Every search run in these program sections will be acknowledged with alarm 16951.

In other words:
When the alarm appears, the user has started a search run (Serupro type) and the search target lies in an area that cannot be searched!

An area that cannot be searched can also be defined implicitly with the machine data $MC_AUTO_IPTR_LOCK.

© Siemens AG, 2002. All rights reserved

Note:
The alarm can only be generated when the simulation is completed during the search run. The alarm cannot be output immediately at the start of the search run.

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
- Alarm display.

Remedy: $MN_SUPPRESS_ALARM_MASK $MC_AUTO_IPTR_LOCK and language command IPTRLOCK

Program Continuation: Clear alarm with the RESET key. Restart part program

17000 Channel %1 block %2 maximum number of symbols exceeded

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The maximum number of symbols defined by machine data 28020 MM_NUM_LUD_NAMES_TOTAL has been exceeded.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Modify machine data
- Reduce the number of symbols (variables, subroutines, parameters)

Program Continuation: Clear alarm with the RESET key. Restart part program

17001 Channel %1 block %2 no memory left for tool/magazine data

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The number of the following tool/magazine data sizes in the NC is given by machine data:
- Number of tools + number of grinding data blocks: 18082 MM_NUM_TOOL
- Number of cutting edges: 18100 MM_NUM_CUTTING_EDGES_IN_TOA

Tools, grinding data blocks, cutting edges can be used independently of the tool management function.

The memory for the following data is available only if the corresponding bit in 18080 MM_TOOL_MANAGEMENT_MASK has been set.
- Number of monitoring data sets: 18100 MM_NUM_CUTTING_EDGES_IN_TOA
- Number of magazines: 18084 MM_NUM_MAGAZINE
- Number of magazine locations: 18086 MM_NUM_MAGAZINE_LOCATION

The following size is determined by software configuration: Number of magazine spacing data blocks: P2 permits 32 such spacing data blocks.

Definition:
- 'Grinding data blocks': Grinding data can be defined for a tool from type 400 to 499. Such a data block occupies additional memory, as it is planned for a cutting edge.
- 'Monitoring data blocks': Each cutting edge of a tool can be supplemented by monitoring data.
- 'Magazine spacing data block': Spacings to other magazines can be defined for magazine locations in internal magazines.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department.
- Modify machine data
- Modify the NC program, i.e. reduce the number of items related to the variable which caused the error condition.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17010 Channel %1 block %2 no memory left

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: When executing/reading files from the active working memory, it was found that there is not enough memory space (e.g. for large multidimensional arrays or when creating tool offset memory).

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Make arrays smaller or make more memory space available for memory management of subroutine calls, tool offsets and user variables (machine data MM_...).
See /FB/, S7 Memory Configuration

Program Continuation: Clear alarm with the RESET key. Restart part program

17020 Channel %1 block %2 illegal array index 1

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A read or write access has been programmed to an array variable with invalid 1st array index. The valid array indices must be contained within the defined array size and the absolute limits (0 - 32 766).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the specification of array elements in the access instruction to match the defined size.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17030 Channel %1 block %2 illegal array index 2

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A read or write access has been programmed to an array variable with invalid 2nd array index. The valid array indices must be contained within the defined array size and the absolute limits (0 - 32 766).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the specification of array elements in the access instruction to match the defined size.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

17040 Channel %1 block %2 illegal axis index

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A read or write access has been programmed to an axial variable in which the axis name cannot be unambiguously imaged on a machine axis.
Example:
Writing of an axial machine data
$MA_... [X]= ... ; but geometry axis X cannot be imaged on a machine axis because of a transformation!

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Deselect transformation before writing the axial data (keyword: TRAFOOF) or use the machine axis names as axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17050 Channel %1 block %2 illegal value

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: On accessing an individual frame element, a frame component other than TRANS, ROT, SCALE or MIRROR was addressed or the function CSCALE has been given a negative scale factor.
Example:
$P_UIFR[5] = CSCALE (X, -2.123)
The frame components are either selected by means of the keywords
TR for translation (TRANS, internal 0)
RT for rotation (ROT, internal 1)
SC for scaling and (SCALE, internal 3)
MI for mirroring (MIRROR, internal 4)
or they are specified directly as an integral value 0, 1, 3, 4.
Example: Access to the rotation around the X axis of the current settable frame.
R10=$P_UIFR[$AC_IFRNUM, X, RT] can also be programmed as:
R10=$P_UIFR[$AC_IFRNUM, X, 1]

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Address frame components only with the keywords provided; program the scale factor between the limits of 0.000 01 to 999.999 99.

Program Continuation: Clear alarm with the RESET key. Restart part program

17055 Channel %1 block %2 GUD variable not existing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The required GUD variable was not found for a MEACALC procedure during read or write access.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Check whether all the GUDs were created for MEACALC.
DEF CHAN INT _MVAR, _OVI[11]
DEF CHAN REAL _OVR[32], _EV[20], _MV[20], _SPEED[4], _SM_R[10], _ISP[3]
DEF NCK REAL _TP[3,10], _WP[3,11], _KB[3,7], _CM[8], _MFS[6]
DEF NCK BOOL _CBIT[16]
DEF NCK INT _CVAL[4].
Program Continuation: Clear alarm with the RESET key. Restart part program

17060 Channel %1 block %2 requested data area too large

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The maximum memory space of 8 KB available for a symbol has been exceeded.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Reduce array dimensions.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17070 Channel %1 block %2 data is write-protected

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An attempt was made to write to a write-protected variable (e.g. a system variable).
Safety Integrated: Safety system variables can only be modified from the safety SPL program.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17080 Channel %1 block %2 %3 value below lower limit

Parameters: %1 = Channel number
%2 = Block number, label
%3 = MD
Definitions: An attempt was made to write a machine data with a value that is smaller than the defined lower limit.
Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.
Remedy: Please inform the authorized personnel/service department. Determine the input limits of the machine data and assign a value within these limits.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17090 Channel %1 block %2 %3 value exceeds upper limit

Parameters: %1 = Channel number
%2 = Block number, label
%3 = MD
Definitions: An attempt was made to write a machine data with a value that is greater than the defined upper limit.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Determine the input limits of the machine data and assign a value within these limits.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17095 Channel %1 block %2 invalid value

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt was made to write an invalid value, e.g. zero, into a machine data.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the value assignment, e.g. a value within the value range not equal to zero.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17100 Channel %1 block %2 digital input/comparator no. %3 not activated

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Input number

Definitions: Either an attempt was made to read a digital input n via the system variable $A_IN[n] and this input has not been activated via NCK machine data 10350 FASTIO_DIG_NUM_INPUTS; or to read a comparator input via system variable $A_INCO[n] and this input belongs to a comparator which has not been activated.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify part program or machine data accordingly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17110 Channel %1 block %2 digital output no. %3 not activated

Parameters: %1 = Channel number
%2 = Block number, label
%3 = No. of output

Definitions: An attempt was made to read or set a digital NCK output (connector X 121) via the system variable $A_OUT [n] with the index [n] greater than the specified upper limit in the NCK machine data 10360 FASTIO_DIG_NUM_OUTPUTS.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program index [n] of the system variable $A_OUT [n] only between 0 and the value in the NCK machine data 10350 FASTIO_DIG_NUM_OUTPUTS.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17120 Channel %1 block %2 analog input no. %3 not activated

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Input number

© Siemens AG, 2002. All rights reserved

Definitions: An attempt has been made by means of the system variable $A_INA[n] to read an analog input n that has not been activated by the MD 10300 FASTIO_ANA_NUM_INPUTS.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify part program or machine data accordingly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17130 Channel %1 block %2 analog output no. %3 not activated

Parameters: %1 = Channel number
%2 = Block number, label
%3 = No. of output

Definitions: An attempt has been made by means of the system variable $A_OUTA[n] to write or read an analog output n that has not been activated by the MD 10310 FASTIO_ANA_NUM_OUTPUTS.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify part program or machine data accordingly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17140 Channel %1 block %2 NCK output %3 is assigned to a function via machine data

Parameters: %1 = Channel number
%2 = Block number, label
%3 = No. of output

Definitions: The programmed digital/analog output is assigned to an NC function (e.g. software cams).

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Use another output or deactivate concurrent NC function via MD.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17150 Channel %1 block %2 maximum of %3 NCK outputs programmable in the block

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Quantity

Definitions: No more than the specified number of outputs may be programmed in an NC block.
The quantity of hardware outputs is defined in the MDs:
10360 FASTIO_DIG_NUM_OUTPUTS and
10310 FASTIO_ANA_NUM_OUTPUTS

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program fewer digital/analog outputs in a block. The specified maximum number applies in each case separately for analog or digital outputs. If necessary, program two NC blocks.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

17160 Channel %1 block %2 no tool selected

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt has been made to access the current tool offset data via the system variables:
$P_AD [n]: Contents of the parameter (n: 1 - 25)
$P_TOOL: Active D number (tool edge number)
$P_TOOLL [n]: Active tool length (n: 1- 3)
$P_TOOLR: Active tool radius
although no tool had been selected previously.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Program or activate a tool offset in the NC program before using the system variables.
Example:
N100 G.. ... T5 D1 ... LF
With the channel-specific machine data:
Modify MD 22550: TOOL_CHANGE_MODE
New tool offset for M function
Modify MD 22560: TOOL_CHANGE_M_CODE
M function with tool change
It is established whether a tool offset is activated in the block with the T word or whether the new offset values are allowed for only when the M word for tool change occurs.

Program Continuation: Clear alarm with the RESET key. Restart part program

17170 Channel %1 block %2 number of symbols too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The predefined symbols could not be read in during power-up.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

17180 Channel %1 block %2 illegal D number

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the displayed block, access is made to a D number (tool edge number) that is not initialized and therefore is not available.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check tool call in the NC part program:
- Correct tool edge number D.. programmed? If no tool edge number is specified, then D1 is automatically active.
- Tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through the V.24 interface.

Description of the system variables $P_DP x [n, m]

© Siemens AG, 2002. All rights reserved

n ... Associated tool number T
m ... Tool edge number D
x ... Parameter number P

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17181 Channel %1 block %2 T no.= %3, D no.= %4 not existing

Parameters: %1 = Channel number
%2 = Block number, label
%3 = T number
%4 = D number

Definitions: A programmed D number was not recognized by the NC. By default, the D number refers to the specified T number. If the flat D number function is active, T= 1 is output.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: If the program is incorrect, remedy the error with a correction block and continue the program. If the data block is missing, download a data block for the specified T/D values onto the NCK (via MMC with overstore) and continue the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17182 Channel %1 block %2 illegal sum correction number

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt was made to access a non-defined total offset of the current tool edge.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Access the total offset memory with $TC_SCP*, $TC_ECP*, check the total offset selection DLx or tool selection Ty or offset selection Dz.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17188 Channel %1 D number %2 defined in tool T no. %3 and %4

Parameters: %1 = Channel number
%2 = Offset number D
%3 = T number of first tool
%4 = T number of second tool

Definitions: The specified D number %2 in the TO unit of channel %1 is not unique. The specified T numbers %3 and %4 each have an offset with number %2. If tool management is active: The specified T numbers belong to tool groups with different names.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: 1. Ensure that the D numbers within the TO unit are unique.
2. If unique numbering is not necessary for subsequent operations, do not use the command.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

17189 Channel %1 D number %2 of tools defined on magazine/location %3 and %4

Parameters: %1 = Channel number
%2 = Offset number D
%3 = Magazine/location number of first tool, '/' as separator
%4 = Magazine/location number of second tool, '/' as separator

Definitions: The specified D number %2 in the TO unit of channel %1 is not unique. The specified T numbers %3 and %4 each have an offset with number %2.
If tool management is active:
The specified T numbers belong to tool groups with different names.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: 1. Ensure that the D numbers within the TO unit are unique, e.g. by renaming the D numbers.
2. If unique numbering is not necessary for subsequent operations, do not use the command.

Program Continuation: Clear alarm with the Delete key or NC START.

17190 Channel %1 block %2 illegal T number

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the displayed block, access is made to a T number (tool number) that is not initialized and therefore not available.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check tool call in the NC part program:
- Correct tool number T.. programmed?
- Tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through the V.24 interface.

Description of the system variables $P_DP x [n, m]
n ... Associated tool number T
m ... Tool edge number D
x ... Parameter number P

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17191 Channel %1 block %2 T= %3 not existing, program %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = T number or T identifier
%4 = Program name

Definitions: A tool identifier which the NCK does not recognize was programmed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: If the program pointer is at an NC block which contains the specified T identifier: If the program is incorrect, remedy the error with a correction block and continue the program. If the data block is missing, create one. You can do this by downloading a data block with all the defined D numbers onto the NCK (via MMC with overstore) and continue the program.
If the program pointer is at an NC block which does not contain the specified T identifier: The error occurred at an earlier point in the program where the T command appeared, but the alarm was not output until the change command was detected.

© Siemens AG, 2002. All rights reserved

If the program is incorrect - T5 programmed instead of T55 - the current block can be corrected with a correction block; i.e. if only M06 is entered, you can correct the block with T55 M06. The incorrect T5 line remains in the program until it is terminated by a RESET or end of program.

In complex program structures with indirect programming, it may not be possible to correct the program. In this case, you can only intervene locally with an overstore block - with T55 in the example. If the data block is missing, create one. You can do this by downloading the data block of the tool with all the defined D numbers onto the NCK (via MMC with overstore), program the T command with overstore, and continue the program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17192 TO unit %1 invalid tool designation of '%2', duplo no. %3. No more replacement tools possible in '%4'.

Parameters: %1 = TO unit

%2 = Tool identifier

%3 = Duplo number

%4 = Group identifier

Definitions: The tool with the specified tool identifier, duplo number cannot accept the group identifier. Reason: The maximum number of replacement tools allowed has already been defined. The name allocation causes the tool to be reallocated to a tool group which already contains the maximum number of replacement tools allowed on this machine.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Use fewer replacement tools or request a different maximum setting from the machine manufacturer.

Program Continuation: Clear alarm with the Delete key or NC START.

17193 Channel %1 block %2 the active tool is no longer on toolholder no./spindle no. %3, program %4

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Toolholder no., spindle no.

%4 = Program name

Definitions: The tool at the specified toolholder/spindle at which the last tool change was carried out as master toolholder or master spindle, has been replaced.

Example:

N10 SETHTH(1)

N20 T="Wz1" ; Tool change at master toolholder 1

N30 SETMTH(2)

N40 T1="Wz2" ; Toolholder 1 is only a secondary toolholder.

Changing the tool does not result in correction deselection.

N50 D5; New correction selection. At present, there is no active tool which D can refer to, i.e. D5 refers to T no. = 0, which results in zero correction.

Reactions: - Interface signals are set.
- Alarm display.

Remedy:
- Modify program:
- Set desired spindle as master spindle or toolholder as master toolholder.
- Then, if required, reset master spindle or master toolholder.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

17194 Channel %1 block %2 no suitable tool found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:
- An attempt was made to access a tool which has not been defined.
- The specified tool does not permit access.
- A tool with the desired properties is not available.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check access to tool:
- Are the parameters of the command correctly programmed?
- Does the status of the tool prevent access?

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17200 Channel %1 block %2 deleting tool data not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt has been made to delete from the part program the tool data for a tool currently being processed. Tool data for tools involved in the current machining operation may not be deleted. This applies both for the tool preselected with T or that has been changed in place of another, and also for tools for which the constant grinding wheel peripheral speed or tool monitoring is active.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Check access to tool offset memory by means of $TC_DP1[t,d] = 0 or deselect tool.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17202 Channel %1 block %2 deleting magazine data not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt was made to delete magazine data which cannot currently be deleted. A magazine with the 'tool in motion' status active cannot be deleted. A tool adapter which is currently allocated to a magazine location cannot be deleted. A tool adapter cannot be deleted if machine data $MN_MM_NUM_TOOL_ADAPTER has the value -1.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: If an attempt to delete a magazine fails
$TC_MAP1[m] = 0 ; Delete magazine with m=magazine no.
$TC_MAP1[0] = 0 ; Delete all magazines
$TC_MAP6[m] = 0 ; Delete magazines and all their tools you must ensure that the magazine does not have the 'tool in motion' status at the time of the call.
If an attempt to delete a tool adapter fails
$TC_ADPTT[a] = -1 ; Delete adapter with number a
$TC_ADPTT[0] = -1 ; Delete all adapters
then the data association with the magazine location or locations must first be canceled with $TC_MPP7[m,p] = 0 ; m = magazine no., p = no. of the location to which the adapter is assigned.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

17210 Channel %1 block %2 access to variable not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The variable cannot be written/read directly from the part program. It is allowed only in motion synchronous actions.
Example for variable:
$P_ACTID (which planes are active)
$AA_DTEPB (axial distance-to-go for reciprocating infeed)
$A_IN (test input)
Safety Integrated: Safety PLC system variables can only be read during the safety SPL startup phase.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17212 Channel %1 tool management: Load manual tool %3, duplo no. %2 onto spindle/ toolholder %4

Parameters: %1 = Channel number
%2 = Duplo no.
%3 = Tool identifier
%4 = Toolholder number (spindle number)

Definitions: Indicates that the specified manual tool must be loaded in the specified toolholder or spindle before the program is continued. A manual tool is a tool whose data are known to the NCK but which is not assigned to a magazine location and is thus not fully accessible to the NCK, and usually also to the machine, for an automatic tool change.

Reactions: - Alarm display.

Remedy: Make sure that the specified tool is loaded in the toolholder. The alarm is cleared automatically after PLC acknowledgement of the tool change on command.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

17214 Channel %1 tool management: remove manual tool %3 from spindle/toolholder %2

Parameters: %1 = Channel number
%2 = Toolholder number (spindle number)
%3 = Tool identifier

Definitions: Indicates that the specified manual tool must be removed from the specified toolholder or spindle before the program is continued. A manual tool is a tool whose data are known to the NCK but which is not assigned to a magazine location and is thus not fully accessible to the NCK, and usually also to the machine, for an automatic tool change.

Reactions: - Alarm display.

Remedy: Make sure that the specified tool is removed from the toolholder. The alarm is cleared automatically after PLC acknowledgement of the tool change on command. Manual tools can only be used efficiently if this is supported by the PLC program.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

17216 Channel %1 tool management: remove manual tool from spindle/toolholder %4 and load manual tool %3, duplo no. %2

Parameters: %1 = Channel number
%2 = Duplo no.
%3 = Tool identifier
%4 = Toolholder number (spindle number)

Definitions: Indicates that the specified manual tool must be loaded in the specified toolholder or spindle before the program is continued and that the manual tool located there must be removed. A manual tool is a tool whose data are known to the NCK but which is not assigned to a magazine location and is thus not fully accessible to the NCK, and usually also to the machine, for an automatic tool change.

Reactions: - Alarm display.

Remedy: Make sure that the manual tools are exchanged. The alarm is cleared automatically after PLC acknowledgement of the tool change on command. Manual tools can only be used efficiently if this is supported by the PLC program.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

17220 Channel %1 block %2 tool not existing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to access a tool via a T no. that has not (yet) been defined. For example, when tools are to be put into magazine locations by programming $TC_MPP6 = 'toolNo'. This is possible only when both the magazine location and the tool given by 'toolNo' have been defined.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17230 Channel %1 block %2 Duplo no. already assigned

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to write a tool Duplo number to the name of which another tool (another T number) already exists with the same Duplo number.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17240 Channel %1 block %2 illegal tool definition

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to modify a tool data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

17250 Channel %1 block %2 illegal magazine definition

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to modify a magazine data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17260 Channel %1 block %2 illegal magazine location definition

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to modify a magazine location data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Correct the NC program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17262 Channel %1 block %2 illegal tool adapter operation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: If an attempt is made to define or cancel a tool adapter assignment with reference to a magazine location and this magazine location already has another tool adapter and/or a tool is located in the adapter or - when canceling an assignment - a tool is still at the location, this alarm will appear. If machine data $MC_MM_NUM_SUMCORR has the value -1, adapters cannot be generated by a write operation to an adapter which is not already defined. While the machine data has this value, you can only write adapter data to adapters which have already been (automatically) assigned to magazine locations.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Assign a maximum of one adapter to a magazine location.
- The magazine location must not contain a tool.
- Machine data $MC_MM_NUM_SUMCORR has value -1: If an alarm occurs when writing one of the system parameters $TC_ADPTx (x=1,2,3,T), the write operation must be modified such that only adapter data which are already associated with the magazine locations are written.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17270 Channel %1 block %2 call-by-reference: illegal variable

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Machine data and system variables must not be transferred as call-by-reference parameters.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Modify NC program: Assign the value of the machine data or of the system variable to a program-local variable and transfer this as parameter.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17500 Channel %1 block %2 axis %3 is not an indexing axis

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: An indexing axis position has been programmed for an axis with the keywords CIC, CAC or CDC that has not been defined as indexing axis in the machine data.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Remove programming instruction for indexing axis positions (CIC, CAC, CDC) from the NC part program or declare the relevant axis to be an indexing axis.
Indexing axis declaration:
Modify MD 30500: INDEX_AX_ASSIGN_POS_TAB
(indexing axis assignment)
The axis will become an indexing axis when an assignment to an indexing position table was made in the stated MD. Two tables are possible (input value 1 or 2).
Modify MD 10900: INDEX_AX_LENGTH_POS_TAB_1
Modify MD 10920: INDEX_AX_LENGTH_POS_TAB_2
(Number of positions for 1st/2nd indexing axis)
Standard value: 0 Maximum value: 60
Modify MD 10910: INDEX_AX_POS_TAB_1 [n]
Modify MD 10930: INDEX_AX_POS_TAB_2 [n]
(Positions of the 1st indexing axis) The absolute axis positions are entered. (The list length is defined via MD 10900).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17501 Channel %1 block %2 indexing axis %3 with Hirth tool system is active

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name

Definitions: The 'Hirth tooth system' function is activated for the indexing axis. This axis can therefore approach only indexing positions, another travel movement of the axis is not possible.

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Correct part program.
Correct FC16 or FC18 call.
Deselect machine data $MA_HIRTH_IS_ACTIVE.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

17502 Channel %1 block %2 indexing axis %3 with Hirth tooth system stop is delayed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name

Definitions: For the indexing axis, the 'Hirth tooth system' function is activated and the override has been set to 0 or another stop condition (e.g. VDI interface signal) is active. Since it is possible to stop only on indexing axes, the next possible indexing position is approached. The alarm is displayed until this position is reached or the stop condition is deactivated.

Reactions: - Alarm display.

Remedy: Wait until the next possible indexing position is reached or set override > 0 or deactivate another stop condition.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

17503 Channel %1 block %2 indexing axis %3 with Hirth tooth system and axis not referenced

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name

Definitions: The 'Hirth tooth system' function is activated for the indexing axis and the axis is to be traversed although it is not referenced.

Reactions: - Alarm display.

Remedy: Reference axis.

Program Continuation: Clear alarm with the Delete key or NC START.

17510 Channel %1 block %2 invalid index for indexing axis %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The programmed index for the indexing axis is beyond the position table range.
Example:
Perform an absolute approach of the 56th position in the list allocated via the axis-specific machine data 30500 INDEX_AX_ASSIGN_POS_TAB with the 1st positioning axis, the number of positions is e.g. only 40 (MD 10900 INDEX_AX_LENGTH_POS_TAB_1 = 40).
N100 G.. U=CAC (56)

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Program the indexing axis position in the NC part program in accordance with the length of the current position table, or add the required value to the position table and adjust the length of the list.

Program Continuation: Clear alarm with the RESET key. Restart part program

17600 Channel %1 block %2 preset on transformed axis %3 not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The displayed axis is involved in the current transformation. This means that is it not possible to set the actual value memory (preset) for this axis.

© Siemens AG, 2002. All rights reserved

Example:
Machine axis A should be set to the new actual value A 100 at the absolute position A 300.
:
N100 G90 G00 A=300
N101 PRESETON A=100

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Avoid preset actual value memory for axes which are participating in a transformation or deselect the transformation with the keyword TRAFOOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17605 Channel %1 block %2 axis %3 transformation active: inhibits rotation of axis container

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The programmed axis/spindle is active in a transformation and the axis container cannot be rotated for this reason.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Modify part program. Deactivate the transformation for this axis/spindle before rotating the axis container or perform the axis container rotation later.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17610 Channel %1 block %2 axis %3 involved in the transformation, action cannot be carried out

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The axis is involved in the active transformation. It can therefore not execute the demanded action, traversing as positioning axis, enable for axis replacement.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Deselect the transformation with TRAFOOF ahead of time or remove the action from the part program block

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17620 Channel %1 block %2 approaching fixed point for transformed axis %3 not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: In the displayed block, an axis is programmed for the fixed point approach (G75) that is involved in the active transformation. Fixed point approach is not performed with this axis!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Remove G75 instruction from the part program block or previously deselect transformation with TRAFOOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17630 Channel %1 block %2 referencing for transformed axis %3 not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: In the displayed block, an axis is programmed for reference point approach (G74) that is involved in the active transformation. Reference point approach is not performed with this axis!

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Remove G74 instruction, or the machine axes involved in transformation, from the part program block or previously deselect the transformation with TRAFOOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17640 Channel %1 block %2 spindle operation for transformed axis %3 not possible

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The axis programmed for the spindle operation is involved in the current transformation as geometry axis. This is not allowed.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: First switch off the transformation function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17650 Channel %1 block %2 machine axis %3 not programmable

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: The machine axis cannot be used in an active transformation. You may be able to program the function in a different coordinate system. For example, it may be possible to specify the retraction position in the basic coordinate system or the workpiece coordinate system. The axis identifier is used to select the coordinate system.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Deactivate the transformation or use another coordinate system.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17800 Channel %1 block %2 illegally coded position programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The position number n specified with the keyword FP=n is not permissible. Two absolute axis positions can be defined as fixed points via the axis-specific MD30 600 FIX_POINT_POS [n].

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Program keyword FP with machine fixed points 1 or 2.
Example:
Approach fixed point 2 with machine axes X1 and Z2.
N100 G75 FP=2 X1=0 Z2=0

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17900 Channel %1 block %2 axis %3 is no machine axis

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: At this point, the block context calls for a machine axis. This is the case with:
- G74 (reference point approach)
- G75 (fixed point approach)

If a geometry or additional axis identifier is used, then it must also be allowed as machine axis identifier (MD 10000 AXCONF_MACHAX_NAME_TAB).

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Use machine axis identifier when programming.

Program Continuation: Clear alarm with the RESET key. Restart part program

18000 Channel %1 block %2 NCK-specific protection zone %3 wrong. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of NCK protection zone
%4 = Error specification

Definitions: There is an error in the definition of the protection zone. The error number gives the specific reason for the alarm. The following meanings apply:
1: Incomplete or conflicting contour definition.
2: Contour encompasses more than one surface area.
3: Tool-related protection zone is not convex.
4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones).
6: Protection zone definition consists of more than 10 contour elements.
7: Tool-related protection zone is defined as inside protection zone.
8: Incorrect parameter used.
9: Protection zone to be activated is not defined.
10: Incorrect modal G code used for protection zone definition.
11: Contour definition incorrect or frame activated.
12: Other errors not specified further.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Modify definition of the protection zone and check MD.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18001 Channel %1 block %2 channel-specific protection zone %3 incorrect. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of the channel-specific protection zone
%4 = Error specification

Definitions: There is an error in the definition of the protection zone. The error number gives the specific reason for the alarm. The following meanings apply:
1: Incomplete or conflicting contour definition.
2: Contour encompasses more than one surface area.
3: Tool-related protection zone is not convex.
4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones).
6: Protection zone definition consists of more than 10 contour elements.
7: Tool-related protection zone is defined as inside protection zone.
8: Incorrect parameter used.
9: Protection zone to be activated is not defined.
10: Incorrect modal G code used for protection zone definition.
11: Contour definition incorrect or frame activated.
12: Other errors not specified further.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department. Modify definition of the protection zone and check MD.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18002 Channel %1 block %2 NCK protection zone %3 cannot be activated. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of NCK protection zone
%4 = Error specification

Definitions: An error has occurred on activating the protection zone. The error number gives the specific reason for the alarm.
The following meanings apply:
1: Incomplete or conflicting contour definition.
2: Contour encompasses more than one surface area.
3: Tool-related protection zone is not convex.
4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones).
6: Protection zone definition consists of more than 10 contour elements.
7: Tool-related protection zone is defined as inside protection zone.
8: Incorrect parameter used.

© Siemens AG, 2002. All rights reserved

9: Protection zone to be activated is not defined.
10: Error in internal structure of the protection zones.
11: Other errors not specified further.
12: The number of protection zones simultaneously active exceeds the maximum number (channel-specific machine data).
13,14: Contour element for protection zones cannot be created.
15,16: No more memory space for the protection zones.
17: No more memory space for the contour elements.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department.
1. Reduce the number of simultaneously active protection zones (MD).
2. Modify part program:
• Delete other protection zones.
• Preprocessing stop.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18003 Channel %1 block %2 NCK protection zone %3 cannot be activated.Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of channel-specific protection zone
%4 = Error specification

Definitions: An error has occurred on activating the protection zone. The error number gives the specific reason for the alarm.
The following meanings apply:
1: Incomplete or conflicting contour definition.
2: Contour encompasses more than one surface area.
3: Tool-related protection zone is not convex.
4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones).
6: Protection zone definition consists of more than 10 contour elements.
7: Tool-related protection zone is defined as inside protection zone.
8: Incorrect parameter used.
9: Protection zone to be activated is not defined.
10: Error in internal structure of the protection zones.
11: Other errors not specified further.
12: The number of protection zones simultaneously active exceeds the maximum number (channel-specific machine data).
13,14: Contour element for protection zones cannot be created.
15,16: No more memory space for the protection zones.
17: No more memory space for the contour elements.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: Please inform the authorized personnel/service department.
1. Reduce the number of simultaneously active protection zones (MD).
2. Modify part program:

© Siemens AG, 2002. All rights reserved

- Delete other protection zones.
- Preprocessing stop.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18004 Channel %1 block %2 orientation of workpiece-related protection zone %3 does not correspond to the orientation of tool-related protection zone %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of workpiece-related protection zone

Definitions: The orientation of the workpiece-related protection zone and the orientation of the tool-related protection zone differ. If the protection zone number is negative, then this is a global protection zone.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Modify the protection zone definition or do not simultaneously activate protection zones that have different orientations.
- Check machine data and modify the protection zone definition if necessary.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18005 Channel %1 block %2 serious error in definition of NCK-specific protection zone %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Protection zone number

Definitions: The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed. This also applies to any that are initiated implicitly such as with G74, M30, M17.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18006 Channel %1 block %2 serious error in definition of channel-specific protection zone %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Protection zone number

Definitions: The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed. This also applies to any that are initiated implicitly such as with G74, M30, M17.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- Correction block is reorganized.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

18100 Channel %1 block %2 invalid value assigned to FXS[]

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The following values are valid at the present time:
0: "Deselect traverse against fixed stop"
1: "Select traverse against fixed stop" valid.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: -

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18101 Channel %1 block %2 invalid value assigned to FXST[]

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Only the range 0.0 - 100.0 is valid at the present time.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: -

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18102 Channel %1 block %2 invalid value assigned to FXSW[]

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Only positive values including zero are valid at the present time.

Reactions:
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy: -

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18200 Channel %1 block %2 curve table: block search stop not allowed with definition CTABDEF

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Program instructions that lead to a preprocessing stop are not allowed within a curve table definition. The system variable $P_CTABDEF can be queried to check whether a table definition is currently active.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Put the block in parenthesis using "IF NOT($P_CTABDEF) ... ENDIF" or remove the instruction that causes the preprocessing stop. Then start the part program again.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

18201 Channel %1 block %2 curve table: table %3 does not exist

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of curve table

Definitions: An attempt was made to use a curve table whose table number is not known in the system \par.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Change the table number in the program instruction or define the curve table with the desired table number.

Program Continuation: Clear alarm with the RESET key. Restart part program

18202 Channel %1 block %2 curve table: instruction CTABEND without CTABDEF not allowed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The CTABEND instruction, which is used to terminate the definition, has been programmed in the program without starting a curve table definition with CTABDEF, or the CTABDEF and CTABEND instructions were not programmed in the same program level.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Remove the CTABEND command or add the CTABDEF(..) command at the appropriate program location. The CTABDEF and CTABEND instructions must be programmed in the same program level (main or subprogram). Start the program again.

Program Continuation: Clear alarm with the RESET key. Restart part program

18300 Channel %1 block %2 frame: fine shift not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Allocation of a fine shift to settable frames or the basic frame is not possible since MD $MN_FRAME_FINE_TRANS is not equal to 1.

Reactions:
- Alarm display.
- Interface signals are set.
- Interpreter stop

Remedy: Please inform the authorized personnel/service department. Modify program or set MD $MN_FRAME_FINE_TRANS to 1.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18310 Channel %1 block %2 frame: illegal rotation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Rotations are not possible with NCU global frames.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

© Siemens AG, 2002. All rights reserved

Remedy: Modify part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

18311 Channel %1 block %2 frame: illegal instruction

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An attempt was made to read or write a frame which does not exist.
Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

18312 Channel %1 block %2 frame: fine shift not configured

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Fine shift must be configured with G58 and G59.
Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Modify machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

18313 Channel %1 block %2 frame: illegal switchover of geometry axes

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: It is not allowed to change the geometry axis assignment because the current frame contains rotations.
Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.
Remedy: Change NC program or set other mode with $MN_FRAME_GEOAX_CHANGE_MODE.
Program Continuation: Clear alarm with the RESET key. Restart part program

18314 Channel %1 block %2 frame: type conflict

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: It is not possible to chain global frames and channel-specific frames. The alarm occurs if a global frame is programmed with a channel axis name and no machine axis on this NCU is assigned to the channel axis.
Channel-specific frames cannot be programmed with machine axis names if there is no corresponding channel axis on this NCU.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

18400 Channel %1 block %2 language change not possible:

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Cause

Definitions: The selection of an external NC language is not possible due to the reason specified. The following reasons are possible (see parameter 3):
1. Invalid machine data settings
2. Active transformation

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Remedy the specified cause of the error before selecting the language.

Program Continuation: Clear alarm with the RESET key. Restart part program

20000 Channel %1 axis %2 reference cam not reached

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: After starting the reference point approach, the rising edge of the reduction cam must be reached within the section defined in the MD 34030 REFP_MAX_CAM_DIST (phase 1 of referencing). (This error occurs only with incremental encoders).

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. There are 3 possible causes of error:
1. The value entered in MD 34030 REFP_MAX_CAM_DIST is too small. Determine the maximum possible distance from the beginning of reference motion up to the reduction cam and compare with the value in the MD: REFP_MAX_CAM_DIST, increase the value in the MD if necessary.
2. The cam signal is not received by the PLC input module. Operate the reference point switch by hand and check the input signal on the NC/PLC interface (route: switch!connector!cable! PLC input!user program).
3. The reference point switch is not operated by the cam. Check the vertical distance between reduction cam and activating switch.

Program Continuation: Clear alarm with the RESET key. Restart part program

20001 Channel %1 axis %2 no cam signal present

Parameters: %1 = Channel number
%2 = Axis name, spindle number

© Siemens AG, 2002. All rights reserved

Definitions: At the beginning of phase 2 of reference point approach, the signal from the reduction cam is no longer available.

Phase 2 of reference point approach begins when the axis remains stationary after deceleration to the reduction cam. The axis then starts in the opposite direction in order to select the next zero marker of the measuring system on leaving the reduction cam or approaching it again (negative/positive edge).

Reactions:
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check whether the deceleration path after the approach velocity is greater than the distance to reference point cam - in which case the axis cannot stop until it is beyond the cam. Use longer cam or reduce the approach velocity in machine data 34020 REFP_VELO_SEARCH_CAM.

When the axis has stopped at the cam, it must be checked whether the signal "DECELERATION REFERENCE POINT APPROACH" is still available at the interface to the NCK (DB 31 - 48, DBX 12.7).

- Hardware: Wire break? Short circuit?
- Software: User program?

Program Continuation: Clear alarm with the RESET key. Restart part program

20002 Channel %1 axis %2 zero mark not found

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: The zero marker of the incremental encoder is not within a defined section.

Phase 2 of reference point approach ends when the zero marker of the encoder has been detected after the rising/falling edge of the PLC interface signal "DECELERATION REFERENCE POINT APPROACH" (DB 31 - 48, DBX 12.7) has given the trigger start. The maximum distance between the trigger start and the zero marker that follows is defined in the machine data 34060 REFP_MAX_MARKER_DIST.

The monitor prevents a zero marker signal from being overtraveled and the next being evaluated as reference point signal. (Faulty cam adjustment or excessive delay by the PLC user program).

Reactions:
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check the cam adjustment and make sure that the distance is sufficient between the end of the cam and the zero marker signal that follows. The path must be greater than the axis can cover in the PLC cycle time.

Increase the machine data 34060 REFP_MAX_MARKER_DIST, but do not select a value greater than the distance between the 2 zero markers. This might result in the monitor being switched off.

Program Continuation: Clear alarm with the RESET key. Restart part program

20003 Channel %1 axis %2 measuring system error

Parameters: %1 = Channel number

%2 = Axis name, spindle number

© Siemens AG, 2002. All rights reserved

Definitions: In a measuring system with distance-coded reference marks, the distance between two adjacent markers has been found to be more than twice the distance entered in the machine data 34300 ENC_REFP_MARKER_DIST.

The control issues the alarm after having made a second attempt in reverse direction with half the traversing velocity and detecting that the distance is too large again.

Reactions:
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Determine the distance between 2 odd reference point markers (reference point marker interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in the machine data 34300 ENC_REFP_MARKER_DIST.

Check the reference point track of the scale including the electronics for the evaluation.

Program Continuation: Clear alarm with the RESET key. Restart part program

20004 Channel %1 axis %2 reference mark missing

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: In the distance-coded length measurement system two reference marks were not found within the defined searching distance (axis-specific MD: 34060 REFP_MAX_MARKER_DIST).

No reduction cam is required for distance-coded scales (but an existing cam will be evaluated). The conventional direction key determines the direction of search. The searching distance 34060 REFP_MAX_MARKER_DIST, within which the two reference point markers are expected is counted commencing at the start point.

Reactions:
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Determine the distance between 2 odd reference point markers (reference point marker interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in the machine data 34060 REFP_MAX_MARKER_DIST.

Check the reference point track of the scale including the electronics for the evaluation.

Program Continuation: Clear alarm with the RESET key. Restart part program

20005 Channel %1 axis %2 reference point approach aborted

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: Channel-specific referencing could not be completed for all stated axes (e.g., abort caused by missing servo enable, measuring system switchover, release of direction key, etc.).

Reactions:
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Check the possible reasons for termination:
- Servo enable missing (DB 31 - 48, DBX 2.1)
- Measuring system switchover (DB 31 - 48, DBX 1.5 and DBX 1.6)
- Traversing key + or missing (DB 31 - 48, DBX 4.6 and DBX 4.7)
- Feed override = 0
- The feed disable is active

The axis-specific MD 34110 REFP_CYCLE_NR determines which axes are involved in the channel-specific referencing.

-1: No channel-specific referencing, NC Start without referencing.

0: No channel-specific referencing, NC Start with referencing.

1-8: Channel-specific referencing. The number entered here corresponds to the referencing sequence. (When all axes with contents 1 have reached the reference point, then the axes with contents 2 start, etc.).

Program Continuation: Clear alarm with the RESET key. Restart part program

20006 Channel %1 axis %2 reference point creep velocity not reached

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: In phase 2 of reference point approach (wait for zero mark), the cam end was reached but the reference point approach velocity was not within the tolerance window. (This can occur when the axis is already at the end of the cam at the beginning of reference point approach. This means that phase 1 has already been concluded and will not be started.)

Phase 2 has been interrupted (this time before the cam) and the reference point traversing will be started once again automatically with phase 1. If the approach velocity is not attained at the second attempt either, the referencing will be aborted with the alarm display.

Approach velocity: 34040 REFP_VELO_SEARCH_MARKER

Velocity tolerance: 35150 SPIND_DES_VELO_TOL.

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Reduce the MD for the approach velocity 34040 REFP_VELO_SEARCH_MARKER and/or increase the MD for the velocity tolerance 35150 SPIND_DES_VELO_TOL.

Program Continuation: Clear alarm with the RESET key. Restart part program

20007 Channel %1 axis %2 reference point approach requires 2 measuring systems

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: 2 encoders are needed for setting 34200 ENC_REFP_MODE = 6!

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Modify reference mode 34200 ENC_REFP_MODE or install and configure a second encoder.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20008 Channel %1 axis %2 reference point approach requires second referenced measuring system

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: When setting 34200 ENC_REFP_MODE = 6 the 2nd encoder must first be referenced.

Reactions:
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Modify referencing mode ENC_REFP_MODE or reference 2nd encoder.

Program Continuation: Clear alarm with the RESET key. Restart part program

20050 Channel %1 axis %2 handwheel mode active

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The axes cannot be traversed in JOG mode using the traversing keys because traversing is still taking place via the handwheel.

Reactions: - Alarm display.

Remedy: Decide whether the axis is to be traversed by means of the jog keys or via the handwheel. End handwheel travel and delete the axial distance-to-go if necessary (interface signal DB 31 - 48, DBX 2.2).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20051 Channel %1 axis %2 handwheel mode not possible

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The axis is already traveling via the traversing keys, so handwheel mode is no longer possible.

Reactions: - Alarm display.

Remedy: Decide whether the axis is to be traversed by means of the jog keys or via the handwheel.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20052 Channel %1 axis %2 already active

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The axis is to traverse as machine axis in JOG mode via the jog keys on the machine control panel. However, this is not possible because:

1. It is already traversing as geometry axis (through the channel-specific interface DB 21 - 28, DBX 12.6, DBX 12.7, DBX 16.6, DBX 16.7 or DBX 20.6 and DBX 20.7) or

2. it is already traversing as machine axis (through the axis-specific interface DB 31 - 48, DBX 4.6 and DBX 4.7) or

3. a frame is valid for a rotated coordinate system and another geometry axis involved in this is already traversing in JOG mode by means of the direction keys.

Reactions: - Alarm display.

Remedy: Stop traversing through the channel or axis interface or stop the other geometry axis.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

20053 Channel %1 axis %2 DRF, FTOCON, external zero point offset not possible

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The axis is traversed in a mode (e.g. referencing) that allows no additional overlaid interpolation.

Reactions: - Alarm display.

Remedy: Wait until the axis has reached its reference position or terminate reference point approach with "Reset" and start DRF once again.

Program Continuation: Clear alarm with the Delete key or NC START.

20054 Channel %1 axis %2 wrong index for indexing axis in JOG mode

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: 1. The displayed indexing axis is to be traversed incrementally in JOG mode (by 1 indexing position). However, no further indexing position is available in the selected direction.
2. The axis is stationary at the last indexing position. In incremental traversing the working area limitation or the software limit switch is reached without an indexing position being located in front of it at which a stop could be made.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.
Correct (add to) the list of indexing positions by means of the machine data
Modify MD 10900: INDEX_AX_LENGTH_POS_TAB_1
Modify MD 10910: INDEX_AX_POS_TAB_1
Modify MD 10920: INDEX_AX_LENGTH_POS_TAB_2
Modify MD 10930: INDEX_AX_POS_TAB_2
or set the working area limits or the software limit switches to other values.

Program Continuation: Clear alarm with the Delete key or NC START.

20055 Channel %1 master spindle not present in JOG mode

Parameters: %1 = Channel number

Definitions: The displayed axis is to be traversed as machine axis in JOG mode with revolutional feed, but no master spindle has been defined from which the actual speed could have been derived.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

Remedy: Please inform the authorized personnel/service department. If the revolutional feed is also to be active in JOG mode, then a master spindle must be declared via the channel-specific machine data 20090 SPIND_DEF_MASTER_SPIND. In this case you have to open a screen in the PARAMETER operating area with the soft keys "SETTINGDATA" and "JOG DATA" and preselect the G function G95 there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feed, the control takes the value assigned in the axis-specific MD 32050 JOG_REV_VELO or in the case of rapid traverse overlay 32040 JOG_REV_VELO_RAPID).
The revolutional feed in JOG mode is deactivated by changing the G function from G95 to G94.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

20056 Channel %1 axis %2 no revolutional feedrate possible. Axis/spindle %3 stationary

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Axis name, spindle number

Definitions: An axis is to travel in JOG with revolutional feed, but the spindle/axis the feed is to be derived from is 0.

Reactions: - Alarm display.

Remedy: Traverse the spindle/axis from which the feed is to be derived.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20057 Channel %1 block %2 revolutional feedrate for axis/spindle %3 is <= zero

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number

Definitions: Revolutional feed has been programmed for an axis/spindle, but the velocity was not programmed or the programmed value is smaller than or equal to zero.

Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- Local alarm reaction.
- Correction block is reorganized.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.
- Correct the part program or
- Specify the correct feed for PLC axes at the VDI interface,
- Specify feed for oscillating axes in the setting data $SA_OSCILL_VELO.

Program Continuation: Clear alarm with the RESET key. Restart part program

20058 Channel %1 axis %2 revolutional feedrate: illegal feed source

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: An axis/spindle is to be traversed at revolutional feedrate. The reference axis/spindle defined in SD 43300 ASSIGN_FEED_PER_REV_SOURCE refers to itself. The coupling caused cannot be executed.

Reactions: - Alarm display.

Remedy: The reference axis/spindle must be modified accordingly in SD 43300.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20060 Channel %1 axis %2 cannot be traversed as geometry axis

Parameters: %1 = Channel number
%2 = Axis name

Definitions: The axis is currently not in "Geometry axis" state. Therefore, it cannot be traversed in JOG mode as geometry axis.
If the abbreviation WCS (workpiece coordinate system) is displayed in the "Position" screen, then only the geometry axes can be traversed by means of the direction keys! (MCS ... Machine coordinate system; all machine axes can now be traversed by using the direction keys on the machine control panel).

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Check the operating steps to establish whether geometry axes really must be traversed, otherwise switch over to the machine axes by activating the "WCS/MCS" key on the machine control panel.
Program Continuation: Clear alarm with the Delete key or NC START.

20061 Channel %1 axis %2 cannot be traversed as orientation axis

Parameters: %1 = Channel number
%2 = Axis name
Definitions: The axis is not an orientation axis and can therefore not be traversed as an orientation axis in JOG mode.
Reactions: - Alarm display.
Remedy: Register the axis as an orientation axis.
Program Continuation: Clear alarm with the Delete key or NC START.

20062 Channel %1 axis %2 already active

Parameters: %1 = Channel number
%2 = Axis name, spindle number
Definitions: The displayed axis is already traversing as a machine axis. Therefore, it cannot be operated as a geometry axis.
Traversing of an axis can take place in JOG mode through 2 different interfaces.
1. as a geometry axis: via the channel-specific interface DB 21 - DB 28, DBX12.6 or DBX12.7
2. as a machine axis: via the axis-specific interface DB 31 - DB 48 DBX8.6 or DBX8.7
With the standard machine control panel, it is not possible to operate an axis as machine axis and geometry axis at the same time.
Reactions: - Alarm display.
Remedy: Do not start the geometry axis until the traversing motion as machine axis has been concluded.
Program Continuation: Clear alarm with the Delete key or NC START.

20063 Channel %1 axis %2 orientation axes cannot be traversed without transformation

Parameters: %1 = Channel number
%2 = Axis name
Definitions: An attempt was made to move an orientation axis in JOG mode without an active orientation transformation.
Reactions: - Alarm display.
Remedy: Activate an orientation transformation.
Program Continuation: Clear alarm with the Delete key or NC START.

20065 Channel %1 master spindle not defined for geometry axes in JOG mode

Parameters: %1 = Channel number
Definitions: The displayed axis is to be traversed as geometry axis in JOG mode with rotary feed, but no master spindle has been defined from which the actual speed could be derived.
Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.

© Siemens AG, 2002. All rights reserved

Remedy: If the revolutional feed is also to be active in JOG mode, then a master spindle must be declared via the channel-specific machine data 20090 SPIND_DEF_MASTER_SPIND. In this case you have to open a screen in the PARAMETER operating area with the soft keys "SETTINGDATA" and "JOG DATA" and preselect the G function G95 there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feed, the control takes the value assigned in the axis-specific MD 32050 JOG_REV_VELO or in the case of rapid traverse overlay 32040 JOG_REV_VELO_RAPID).

The revolutional feed in JOG mode is deactivated by changing the G function from G95 to G94.

Program Continuation: Clear alarm with the Delete key or NC START.

20070 Channel %1 axis %2 programmed end position is behind software limit switch %3

Parameters: %1 = Channel number

%2 = Axis number

%3 = "+" or "-"

Definitions: The axis is traversed as a concurrent positioning axis by the PLC and the target position is situated behind the corresponding software limit switch. The axis is not traversed.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Specify smaller target position. Modify MD for SW limit switch. Possibly activate another SW limit switch.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20071 Channel %1 axis %2 programmed end position is behind working area limit %3

Parameters: %1 = Channel number

%2 = Axis number

%3 = "+" or "-"

Definitions: The displayed axis is operated as a concurrent positioning axis. Its target position is behind the preset working area limitation. The axis is not traversed.

Reactions: - Alarm display.

Remedy:
- Specify smaller target position.
- Deactivate working area limitation.
- Set working area limitation differentially.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20072 Channel %1 axis %2 is not an indexing axis

Parameters: %1 = Channel number

%2 = Axis number

Definitions: The displayed axis is operated as a concurrent positioning axis. Its target position is parameterized in the FC INDEX-AXIS as indexing position number, but the axis is not an indexing axis.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. The FC POS-AXIS for linear and rotary axes should be used or the axis should be declared as an indexing axis. Corresponding machine data for indexing axis declaration:

Modify MD 30500: INDEX_AX_ASSIGN_POS_TAB

Modify MD 10900: INDEX_AX_LENGTH_POS_TAB_1

Modify MD 10910: INDEX_AX_POS_TAB_1

Modify MD 10920: INDEX_AX_LENGTH_POS_TAB_2

Modify MD 10930: INDEX_AX_POS_TAB_2

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

20073 Channel %1 axis %2 cannot be repositioned

Parameters: %1 = Channel number
%2 = Axis number

Definitions: The concurrent positioning axis cannot be positioned because it has already been restarted via the VDI interface and is still active. No repositioning motion takes place and the motion initiated by the VDI interface is not affected.

Reactions: - Alarm display.

Remedy: None.

Program Continuation: Clear alarm with the Delete key or NC START.

20074 Channel %1 axis %2 wrong index position

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: For a concurrent positioning axis declared as indexing axis, the PLC has given an index number that is not available in the table.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Check the indexing axis number given by the PLC and correct this if necessary. If the indexing axis number is correct and the alarm results from an indexing position table that has been set too short, check the machine data for indexing axis declaration.
Modify MD 30500: INDEX_AX_ASSIGN_POS_TAB
Modify MD 10900: INDEX_AX_LENGTH_POS_TAB_1
Modify MD 10910: INDEX_AX_POS_TAB_1
Modify MD 10920: INDEX_AX_LENGTH_POS_TAB_2
Modify MD 10930: INDEX_AX_POS_TAB_2

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20075 Channel %1 axis %2 can currently not oscillate

Parameters: %1 = Channel number
%2 = Axis number

Definitions: The axis cannot perform an oscillating movement now because it is already being traversed, e.g. in JOG mode.

Reactions: - Alarm display.

Remedy: End the other traversing motion.

Program Continuation: Clear alarm with the Delete key or NC START.

20076 Channel %1 axis %2 oscillating - mode change not possible

Parameters: %1 = Channel number
%2 = Axis number

Definitions: The axis is performing an oscillating movement. Mode change is not possible because oscillation is not allowed in the selected mode.

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Do not initiate mode change. Cause the PLC to check the axis and make sure in the PLC program that the axis ends oscillation if such mode changes take place.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20077 Channel %1 axis %2 programmed position is behind software limit switch %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"

Definitions: The axis is traversed as an oscillating axis and the target position (reversal position or end position) is located behind the corresponding software limit switch. The axis is not traversed.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Specify smaller target position.
Modify MD for SW limit switch.
Possibly activate another SW limit switch.

Program Continuation: Clear alarm with the RESET key. Restart part program

20078 Channel %1 axis %2 programmed position is behind working area limit %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"

Definitions: The axis is traversed as an oscillating axis and the target position (reversal position or end position) is located behind the corresponding valid working area limitation. The axis is not traversed.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Specify smaller target position.
Deactivate working area limitation.
Set working area limitation differentially.

Program Continuation: Clear alarm with the RESET key. Restart part program

20079 Channel %1 axis %2 oscillation path length %3 <= 0

Parameters: %1 = Channel number
%2 = Axis number
%3 = Length

Definitions: The axis is traversed as an oscillating axis and the distance to be traversed is smaller than or equal to zero.
For example, both reversal points are situated on an identical position, one reversal point was shifted against the oscillating direction beyond the other reversal point. The axis is not traversed.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Start disable in this channel.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Specify correct target position (reversal position, end position).

Program Continuation: Clear alarm with the RESET key. Restart part program

20080 Channel %1 axis %2 no handwheel assigned for overlaid motion

Parameters: %1 = Channel number
%2 = Axis number

Definitions: No handwheel has been assigned for this specified axis after handwheel overlay has been started in automatic mode. If the axis identifier is missing in the alarm with active velocity overlay FD > 0, then the 1st geometry axis has not been defined in the NC channel. In this case the block is executed without handwheel control.

Reactions: - Alarm display.

Remedy: If handwheel control is required, a handwheel must be activated.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20085 Channel %1 contour handwheel: traverse direction or overtravel of beginning of block not allowed

Parameters: %1 = Channel number

Definitions: Travel takes place on the path with the contour handwheel in the opposite direction to the programmed travel direction and the starting point of the path has been reached at the start of the block.

Reactions: - Alarm display.

Remedy: Turn the contour handwheel in the opposite direction.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20090 Axis %1 travel to fixed stop not possible. Check programming and axis data.

Parameters: %1 = Axis name, spindle number

Definitions: 1. The "Traverse against fixed stop" function has been programmed with FXS[AX]=1 but the axis does not (yet) support this. Check MD 37000. This function is not available for gantry axes and simulated axes.

2. On selection, no movement was programmed for axis AX. AX is a machine axis identifier.

3. It is always necessary to program a traversing movement in the selection block for the axis/spindle for which the "Traverse against fixed stop" function is activated.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
- Check the axis type.
- Check MD 37000.
- Is a machine axis movement missing in the approach block?

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20091 Axis %1 has not reached fixed stop

Parameters: %1 = Axis name, spindle number

Definitions: On attempting to traverse against a fixed stop, the programmed end position has been reached or the traversing movement has been aborted. The alarm can be concealed by means of the machine data $MA_FIXED_STOP_ALARM_MASK.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct the part program and the settings:
- Has the traversing block been aborted?
- If the axis position does not correspond to the programmed end position, then correct the end position.
- If the programmed end position is in the part, the triggering criterion must be checked.
- Has the contour deviation leading to triggering been dimensioned too large? Has the torque limit been set too high?

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20092 Axis %1 travel to fixed stop still active

Parameters: %1 = Axis name, spindle number

Definitions: An attempt has been made to move an axis while it is in fixed stop or while the deselection function has not yet been completed.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Check the following:
- Has the axis at the fixed stop also been moved by a traversing movement of geometry axes?
- Is a selection carried out even though the axis is stationary at the stop?

© Siemens AG, 2002. All rights reserved

- Has the deselection process been interrupted by a RESET?
- Has the PLC switched the acknowledgement signals?

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20093 Axis %1 standstill monitoring at fixed-stop end point has been triggered

Parameters: %1 = Axis name, spindle number

Definitions: The position of the axis has been beyond the zero speed window ever since selection has been completed.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Check the mechanical components, e.g. has the stop broken away? Has the part to be clamped given way?
- Position window for zero speed control too small (37020 MD: $MA_FIXED_STOP_WINDOW_DEF) (43520 setting data: $SA_FIXED_STOP_WINDOW). Default is 1 mm in each case.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20094 Axis %1 function has been aborted

Parameters: %1 = Axis name, spindle number

Definitions: The function has been aborted. The possible reasons for this are:

- Because a pulse disable has occurred, the torque can no longer be provided.
- The PLC has removed the acknowledgments.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check whether

- there is a pulse disable from the infeed/regenerative-feedback unit or from the PLC?
- the acknowledgement bits have been deleted by the PLC even though NCK has not requested deselection?

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20095 Axis %1 illegal torque, current torque %2

Parameters: %1 = Axis name, spindle number

%2 = Current holding torque when brake test selected

© Siemens AG, 2002. All rights reserved

Definitions: The current holding torque, when brake test selected, cannot be attained with the present parameterization of the brake test.

Reactions: - Alarm display.

Remedy: Check the parameterization for the brake test function check:
- The torque for the counterweight in the drive machine data 1192 should be nearly the same as the current holding torque. The current holding torque is displayed in the alarm text.
- The torque set for the MA_BRAKETEST_TORQUE must be greater than the current holding torque.

Program Continuation: Clear alarm with the Delete key or NC START.

20096 Axis %1 brake test aborted, additional information %2

Parameters: %1 = Axis name, spindle number

%2 = Error information based on $VA_FXS_INFO

Definitions: The brake test has detected a problem. The additional info provides more detailed information on the cause of the alarm. The explanation can be found in the $VA_FXS_INFO system variable documentation.

Additional information:

0: No additional information available.

1: Axis type is not a PLC or command axis.

2: End position reached, motion completed.

3: Abort by NC RESET (key reset).

4: Moved out of monitoring window.

5: Torque reduction rejected by drive.

6: PLC has cancelled enables.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Note the supplementary conditions of the brake test, see additional info.

Program Continuation: Clear alarm with the Delete key or NC START.

20100 Channel %1: invalid configuration for digitizing function

Parameters: %1 = Channel number

Definitions:
- The digitizing function requires 3 geometry axes to be defined in the channel.
- With the baud rate available for transferring the actual positions and setpoint velocities between NC and digitizing unit, the interpolation cycle cannot be set to a value of less than 5 ms.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Define 3 geometry axes for the digitizing channel by means of machine data.
- Use interpolation cycle of greater than 5 ms.

Program Continuation: Switch control OFF - ON.

20101 Communication with the digitizer not possible

Definitions: The attempt to synchronize the communications link to the digitizing unit and to transfer the machine parameters was aborted after the preset timeout limit of 15 seconds was exceeded.

Reactions: - Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Check the connection to the digitizing unit (RS422 cable, supply voltage) and whether the digitizing unit is switched on.
Program Continuation: Clear alarm with the Delete key or NC START.

20102 Channel %1: No or invalid trafo at digitizing active

Parameters: %1 = Channel number
Definitions: Prerequisite for the 3+2 axis digitizing is an active kinematic transformation. Permitted transformations are the general 5-axis transformation and the universal inclinable head.
Reactions: - Alarm display.
- Interface signals are set.
Remedy: • Before digitizing, activate a permitted transformation.
• Select 3-axis mode for the digitizing via machine data.
Program Continuation: Clear alarm with the Delete key or NC START.

20103 Channel %1: digitizing module does not support 3+2 axis digitizing

Parameters: %1 = Channel number
Definitions: Prerequisite for 3+2 axis digitizing is that the NCU and the digitizing module both have the 3+2 axis mode.
Reactions: - Alarm display.
- Interface signals are set.
Remedy: • SW update for the digitizing module.
• Select 3-axis mode for the digitizing via machine data.
Program Continuation: Clear alarm with the Delete key or NC START.

20105 Channel %1: axes stopped by digitizer. Error code: %2

Parameters: %1 = Channel number
%2 = Error code of digitizing unit
Definitions: The digitizing unit has recognized an error in the communication and signaled this to the NC.
Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
Remedy: Please inform the authorized personnel/service department. Error code 1: Check cable connection leading to the digitizing unit. Other error codes: See manual for digitizing unit.
Program Continuation: Clear alarm with the RESET key. Restart part program

20106 Emergency stop set by the digitizer

Definitions: The digitizing unit has recognized a serious error and triggered an emergency stop. Cause: See display on the digitizing unit.
Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
Remedy: -
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20108 Invalid data package received from the digitizer. Error codes: %1, %2

Parameters: %1 = Error code of cyclic packet
%2 = Error code of out-of-band packet

Definitions: A data packet received by the digitizing unit could not be evaluated.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Error code: 0, 0: Check cable connection leading to the NC. Other error codes: e.g. wrong header, incorrect checksum (development documentation).

Program Continuation: Clear alarm with the RESET key. Restart part program

20109 Error in communication with the digitizer: status code of com-circuit: %1

Parameters: %1 = Status byte

Definitions: The circuit for serial communication with the digitizing unit signals a transmission error via its status byte (framing error, parity etc.).

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Check connection cable leading to the digitizing unit: In particular screening.

Program Continuation: Clear alarm with the RESET key. Restart part program

20120 Axis %1: too many compensation relations

Parameters: %1 = Axis name, spindle number

Definitions: IInterpolatory compensation with tables. For each axis, the maximum number of compensation relationships defined may be no more than the number of axes in the system. In this alarm, the interpolatory compensation in the axis is switched off automatically.

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Check table parameters $AN_CEC_OUTPUT_AXIS and correct and/or switch off one or more tables ($SN_CEC_TABLE_ENABLE).

Program Continuation: Clear alarm with the RESET key. Restart part program

20121 Axis %1: Configuration error in compensation table %2

Parameters: %1 = Axis name, spindle number
%2 = Compensation table

Definitions: IInterpolatory compensation with tables. The settings for the specified table are not allowed. $AN_CEC_MAX >= $AN_CEC_MIN and $AN_CEC_STEP != 0 apply to system variables. This table is switched off automatically.

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and correct the characteristic data in the compensation table. If the error cannot be found, the alarm can be suppressed by switching off the table ($SN_CEC_TABLE_ENABLE) or switching off compensation in the axis ($MA_CEC_ENABLE).

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20122 Compensation table %1: invalid axis assignment

Parameters: %1 = Compensation table

Definitions: IInterpolatory compensation with tables. The input or output axes assignment in the given table is not allowed. $AN_CEC_INPUT_AXIS and $AN_CEC_OUTPUT_AXIS != 0 apply to system variables. This table is automatically switched off.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and correct the axis assignment in the compensation table. If the error cannot be found, the alarm can be suppressed by switching off the table ($SN_CEC_TABLE_ENABLE) or switching off compensation in the axis ($MA_CEC_ENABLE).

Program Continuation: Clear alarm with the RESET key. Restart part program

20123 Axis %1: different output assignment of multiplied tables

Parameters: %1 = Axis name, spindle number

Definitions: IInterpolatory compensation with tables. The two tables whose outputs are to be multiplied together have different output axes assigned to them. The compensation in this axis is automatically switched off.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and correct the characteristic data in the compensation table ($AN_CEC_OUTPUT_AXIS and $AN_CEC_MULT_BY_TABLE).

If the error cannot be found, the alarm can be suppressed by switching off the compensation in the axis ($MA_CEC_ENABLE) or the tables, ($SN_CEC_TABLE_ENABLE).

Program Continuation: Clear alarm with the RESET key. Restart part program

20124 Axis %1: sum of compensation values too large

Parameters: %1 = Axis name, spindle number

Definitions: The sum of the compensation values from all tables assigned to the axis had exceeded the limit value $MA_CEC_MAX_SUM and had to be limited. Contour errors could have occurred as a result.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Check characteristic data of the compensation tables assigned to the axis.

Check characteristic curves in the tables ($AN_CEC).

Program Continuation: Clear alarm with the RESET key. Restart part program

20125 Axis %1: change of compensation value is too rapid

Parameters: %1 = Axis name, spindle number

Definitions: The compensation value has changed more rapidly than has been allowed for in 32730 CEC_MAX_VELO. It had to be limited temporarily. The missing section is repeated later but contour errors might have occurred.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Check characteristic data of the compensation tables assigned to the axis.

Check characteristic curves in the tables ($AN_CEC). Possibly one of the input axes has moved more rapidly than provided for.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

20130 Channel %1 contour tunnel monitoring

Parameters: %1 = Channel number

Definitions: The tool tip has exited the tunnel placed around the desired contour, i.e. the distance between tool tip and desired contour was greater than specified in the MD 21050 CONTOUR_TUNNEL_TOL.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the following points in turn:

1. Is the machine in working order? That is, has the alarm been tripped by a sluggish axis, tool breakage or collision?

2. If the machine is in working order, reduce the velocity or improve the controller setting.

3. Possibly increase the size of the tunnel and monitor errors via analog output in order to ascertain the cause.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

20140 Channel %1 motion synchronous action: traversing of command axis %2 see NC alarm %3

Parameters: %1 = Channel number

%2 = Axis

%3 = NC alarm

Definitions: An NC alarm was detected for a command axis which is to be traversed from a synchronous action. The NC alarm is indicated by an MMC alarm number in the 3rd parameter.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: See help information for the additional alarms.

Program Continuation: Clear alarm with the RESET key. Restart part program

20141 Channel %1 motion synchronous action: illegal axis type

Parameters: %1 = Channel number

Definitions: The requested command is not permissible in the current axis status for the command axis or spindle. This alarm occurs with command axes (POS, MOV), spindle commands from motion synchronous actions (M3/M4/M5, SPOS), coupled motion (TRAILON, TRAILOF) and lead value coupling (LEADON, LEADOF).

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: First stop the axis or deactivate the coupling, then select a new status.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20142 Channel %1 command axis %2: rotation of axis container already enabled

Parameters: %1 = Channel number
%2 = Axis

Definitions: The synchronized action instruction is not allowed on a spindle enabled for the axis container rotation. The alarm only occurs if the spindle is handed to another NCU.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Initiate the synchronized action instruction before the axis container rotation enable or after the end of the rotation (depending on the application).

Program Continuation: Clear alarm with the RESET key. Restart part program

20143 Channel %1 axis %2 command axis cannot be started as it is controlled by the PLC

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: An attempt has been made to start a command axis by means of a block-related or modal synchronous action. This start is not possible as the axis is controlled by the PLC.

Reactions:
- Alarm display.

Remedy: End control of the axis by the PLC and therefore return it to the channel or start the command axis with a static synchronous action.

Program Continuation: Clear alarm with the Delete key or NC START.

20144 Channel %1 block %2 motion synchronous action: system variable access not possible

Parameters: %1 = Channel number
%2 = Block number

Definitions: When using system variables, it is assumed that a read/write operation can access the required data successfully. In accesses to encoder actual values or digital I/Os, the result depends on the availability of the corresponding hardware components. If an access within synchronized actions does not return a valid value, alarm 20144 is output. Outside synchronized actions, such a read/write access causes block execution to be interrupted until the result is available. Block execution is subsequently continued.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Before reading/writing system variables, ensure that it is possible to access the required hardware components.

Program Continuation: Clear alarm with the RESET key. Restart part program

20145 Channel %1 block %2 motion synchronous action: arithmetic error

Parameters: %1 = Channel number
%2 = Block number

Definitions: In calculating an arithmetic expression for a motion synchronous action, an overflow has occurred (e.g. division by zero).

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Correct error in expression.
Program Continuation: Clear alarm with the RESET key. Restart part program

20146 Channel %1 block %2 motion synchronous action: nesting depth exceeded

Parameters: %1 = Channel number
%2 = Block number
Definitions: For calculating arithmetic expressions in motion synchronous blocks, an operand stack with a fixed set size is used. With very complex expressions, this stack can overflow.
Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
Remedy: Correct error in expression.
Program Continuation: Clear alarm with the RESET key. Restart part program

20147 Channel %1 block %2 motion synchronous action: command not executable

Parameters: %1 = Channel number
%2 = Block number
Definitions: One of the commands for the synchronous action block cannot be executed, e.g. it is not possible to perform a Reset to the synchronous action.
Measurement level 2
- Embargo version does not allow measurement from a synchronized action
- MEASA was programmed in a synchronized action
- Measurement is already active
- Programming error (see alarm 21701)

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
Remedy: Change synchronous action.
Measurement level 2
Execute the measurement task from an NC program first, in order to improve the error diagnostics. Only include it in the synchronized action when the first error-free run has been performed.
Program Continuation: Clear alarm with the RESET key. Restart part program

20148 Channel %1 block %2 motion synchronous action: internal error %3

Parameters: %1 = Channel number
%2 = Block number
%3 = Error code
Definitions: An internal error has occurred during processing of a synchronous action. The error code is for diagnostics purposes. Please make a note and contact the manufacturer.
Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
Remedy: Change synchronous action.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

20149 Channel %1 block %2 motion synchronous action: illegal index

Parameters: %1 = Channel number
%2 = Block number

Definitions: An invalid index was used for access to a variable in motion synchronous action.
Example: ... DO $R[$AC_MARKER[1]] = 100
This error occurs if the value of marker 1 is greater than the maximum permissible R parameter number.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Use a valid index.

Program Continuation: Clear alarm with the RESET key. Restart part program

20150 Channel %1 tool management: PLC terminates interrupted command

Parameters: %1 = Channel number

Definitions: Indication that the PLC has terminated an interrupted command (with alarm output) from the tool management - tool change.

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: For information only.

Program Continuation: Clear alarm with the Delete key or NC START.

20160 Channel %1 tool management: PLC can terminate only incorrectly aborted commands

Parameters: %1 = Channel number

Definitions: Indication that the PLC wanted to interrupt an active command from the tool management (tool change); or that there is no command active for abort. NCK refuses because the channel status is either 'active' (abort is then not allowed), or 'reset' (then there is nothing to abort).

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: For information only.

Program Continuation: Clear alarm with the Delete key or NC START.

20170 Channel %1 machine data $AC_FIFO invalid

Parameters: %1 = Channel number

Definitions: The structure of the FIFO variable $AC_FIFO - $AC_FIFO10 determined by the machine data $MC_NUM_AC_FIFO, $MC_START_AC_FIFO, $MC_LEN_AC_FIFO and $MC_MODE_AC_FIFO cannot be stored in the R parameter field defined in $MC_MM_NUM_R_PARAM.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Increase the number of the R parameters or reduce the FIFO elements.
$MC_MM_NUM_R_PARAM = $MC_START_AC_FIFO + $MC_NUM_AC_FIFO x ($MC_LEN_AC_FIFO + 6)

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

20200 Channel %1 invalid spindle number %2 with tool fine compensation

Parameters: %1 = Channel number target channel
%2 = Spindle number

Definitions: There is no spindle/axis assignment in the target channel for the spindle specified in the PUTFTOC command.

Reactions:
- Alarm display.
- Interpreter stop
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Modify program in channel that writes the tool fine compensation.

Program Continuation: Clear alarm with the RESET key. Restart part program

20201 Channel %1 spindle %2 no tool assigned

Parameters: %1 = Channel number
%2 = Spindle number

Definitions: In order to make allowance for the fine tool compensation for the tool currently in the spindle, a spindle/tool assignment must be active. This is not presently the case for the programmed spindle in the target channel of fine tool compensation.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy:
1. Modify the part program (write the tool fine compensation).
2. Establish spindle/tool assignment by programming:
   - TMON (tool monitoring)
   - GWPSON (tool selection)

Program Continuation: Clear alarm with the RESET key. Restart part program

20203 Channel %1 no active tool

Parameters: %1 = Channel number

Definitions: A tool fine compensation has been written for the active tool of channel %1 with PUTFTOC. No tool is active in this channel. Therefore, the compensation cannot be assign-ed.

Reactions:
- Alarm display.
- Interpreter stop
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct the program.

Program Continuation: Clear alarm with the RESET key. Restart part program

20204 Channel %1 PUTFTOC command not allowed with FTOCOF

Parameters: %1 = Channel number

Definitions: A tool fine compensation has been written for channel %1 with PUTFTOC. The tool fine compensation is not active in this channel. FTOCON must be active in the target channel of the PUTFTOC command.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interpreter stop
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Correct the program in the machining channel: Select FTOCON so that the channel is ready to receive the PUTFTOC command.

Program Continuation: Clear alarm with the RESET key. Restart part program

20210 Channel %1 block %3 spindle %2 wrong values for centerless grinding

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: It was not possible to calculate a tool diameter (no speed specified for the spindle) for centerless grinding because it was not allowed by the input positions. The old S value still applies.

Reactions: - Alarm display.

Remedy:
- Modify program
- Select new traversing positions for centerless axes
- or suppress computation by G00.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

20211 Channel %1 block %3 spindle %2 support point beyond range limits

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: The support point calculated for centerless grinding is beyond the range limits.
Machine data:
Modify MD 21518: TRACLG_CONTACT_UPPER_LIMIT
Modify MD 21520: TRACLG_CONTACT_LOWER_LIMIT

Reactions: - Alarm display.

Remedy:
- Check centerless axis positions and machine data.
- Modify program.
- Select new traversing positions for centerless axes
- or suppress computation by G00.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

21600 Monitoring for ESR active

Definitions: -

Reactions: - Alarm display.
- NC not ready.
- All alarm reactions are delayed by one IPO cycle with this alarm.

Remedy: The display can be suppressed with the machine data MD 11410: SUPPRESS_ALARM_MASK Bit 16 = 1

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

21610 Channel %1 axis %2 encoder %3 frequency limit exceeded

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = String (encoder number)

Definitions: The maximum permissible frequency of the currently active encoder (axis-specific interface signal DB 31 - 48, DBX 1.5 and DBX 1.6) in the axis-specific machine data 36 300 ENC_FREQ_LIMIT [n] (n ... encoder number, 1 or 2) has been exceeded. The reference of the actual value to the mechanical carriage position may be lost.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Modify MD 36300: ENC_FREQ_LIMIT [n] and interface signal position measuring system 1/2 (DB 31 - 48, DBX 1.5 and DBX 1.6).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

21611 Channel %1 NC-controlled Extended Stop/Retract triggered

Parameters: %1 = Channel number

Definitions: "NC-controlled Extended Stop/Retract" triggered.

Reactions:
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- All channel-specific alarm reactions are delayed with this alarm, alarm display.

Remedy: Reset

Program Continuation: Clear alarm with the RESET key. Restart part program

21612 Channel %1 axis %2 VDI signal 'Servo enable' reset during motion

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The interface signal "Servo enable" (DB31 - 48, DBX 2.1) has been set to 0 for the displayed axis even though one of the axes in the geometry grouping was in motion.
The axes entered in the channel-specific MD array 20050 AXCONF_GEOAX_ASSIGN_TAB count as axes belonging to the geometry grouping. Servo enable must exist for all available geometry axes, regardless of whether they are currently in motion or not.
Occurs in connection with SAFETY function: If a test stop is performed with linked axes, the alarm is issued if a motion command from the ELT grouping is pending during the test stop of the slave axis.

© Siemens AG, 2002. All rights reserved

Reactions: - NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check the interface signal "Servo enable" (DB31 - 48, DBX 2.1, e.g. with the PLC status display in the DIAGNOSTICS operating area). Trace back the signal to the sections in the PLC user program at which it is linked and set/reset.

With SAFETY: With active actual-value linkage, the output of the error message on the slave axis can be prevented by increasing MD 36060 $MA_STANDSTILL_VELO_TOL to 100 and 200 (default value is 5 mm).

Program Continuation: Clear alarm with the Delete key or NC START.

21613 Axis %1 measuring system changing

Parameters: %1 = Axis name, spindle number

Definitions: The measuring system for this axis is changing.

Reactions: - Alarm display.

Remedy: -

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

21614 Channel %1 axis %2 hardware limit switch %3

Parameters: %1 = Channel number

%2 = Axis name, spindle number

%3 = String (+, - or +/-)

Definitions: The VDI signal "Hardware limit switch" (DB 31 - 48, DBX 12.0 or DBX 12.1) has been set at the NC/PLC interface.

Reactions: - Alarm display.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department.

1. With axes that have already been referenced, the software limit switch 1 or 2 should respond before the hardware limit switch is reached. Check MD 36110 POS_LIMIT_PLUS, 36100 POS_LIMIT_MINUS, 36130 POS_LIMIT_PLUS2 and 36120 POS_LIMIT_MINUS2 and the interface signal for selection of 1st/2nd software limit switch (DB 31 - 48, DBX 12.2 and 12.3) and correct if necessary (PLC user program).

2. If the axis has not yet been moved to the reference point, it is possible to depart from the hardware limit switch in the opposite direction in JOG mode.

3. Check PLC user program and the connection from the switch to the PLC input module, provided the axis has not yet reached the hardware limit switch at all.

Program Continuation: Clear alarm with the RESET key. Restart part program

21615 Channel %1 axis %2 taken from traverse mode to follow-up mode

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: This axis has been taken from traverse mode and put into "Follow-up" mode, for instance because the pulse enable for the drive has been reset.

© Siemens AG, 2002. All rights reserved

Reactions: - NC Start disable in this channel.
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

21616 Channel %1 block %2 overlaid motion active at transformation switchover

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The overlaid motion in the BCS changes its significance because of the transformation change and can therefore lead to undesired axis movements.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.

Remedy: Take out the overlaid movement.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

21617 Channel %1 block %2 transformation does not allow to traverse the pole

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The preset curve passes through the pole or a forbidden area of the transformation.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Modify part program (if alarm has occurred in AUTO mode).
To escape from the alarm position, transformation must be deselected (it is not enough to try a RESET if the transformer remains active when RESET is applied).

Program Continuation: Clear alarm with the RESET key. Restart part program

21618 Channel %1 as from block %2 transformation active: overlaid motion too great

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The share of overlaid motion on the transformation-related axes is so high that the path movement planned by the preparation no longer sufficiently corresponds to the actual ratio for the interpolation. Strategy of singularities, monitoring of working range limitation and dynamic Look Ahead are possibly no longer correct.

Reactions: - Alarm display.

Remedy: With overlaid motion it is necessary to keep a sufficiently large path safety distance with regard to poles and working range limitations.

Program Continuation: Clear alarm with the Delete key or NC START.

21619 Channel %1 block %2 transformation active: motion not possible

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The machine kinematics does not allow the specified motion. Transformation-dependent error causes can be in:

TRANSMIT: A (circular) area exists around the pole, where positioning is not possible. The area is caused by the fact that the tool reference point cannot be traversed as far as into the pole. The area is defined by:

- the machine data ($MC_TRANSMIT_BASE_TOOL..)
- the active tool length compensation (see $TC_DP..).

Whether the tool length compensation is included in the calculation depends on the working plane selected (see G17,..). The machine stops at the edge of the area where positioning is not possible.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Modify part program. Change the incorrectly specified tool length compensation.

Note: RESET alone is not enough if transformation also remains active during RESET.

Program Continuation: Clear alarm with the RESET key. Restart part program

21650 Channel %1 axis %2 overlaid motion not allowed

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Definitions: An overlaid motion was requested for the axis, however, this is not allowed due to the machine data FRAME_OR_CORRPOS_NOTALLOWED.

Reactions: - Alarm display.
- Local alarm reaction.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Deselect the overlaid motion or change machine data FRAME_OR_CORRPOS_NOTALLOWED.

Program Continuation: Clear alarm with the RESET key. Restart part program

21660 Channel %1 block %2 axis %3 conflict between SYNACT: $AA_OFF and CORROF

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Axis name

Definitions: When deselecting the position offset ($AA_OFF) via the part program command CORROF (<axis>, "AA_OFF") an active synchronized action is detected that immediately sets $AA_OFF for the axis (DO_$AA_OFF [<axis>] =<value>). Deselection is executed and $AA_OFF not set again.

Reactions: - Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

© Siemens AG, 2002. All rights reserved

21665 Channel %1 $AA_TOFF cleared

Parameters: %1 = Channel number

Definitions: If the tool position is changed with RESET and $AA_TOFF is active during RESET, the position offset ($AA_TOFF) is cleared.

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy: Modify the RESET setting in $AA_TOFF_MODE.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

21670 Channel %1 block %2 illegal change of tool direction with $AA_TOFF active

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: If an offset has been activated in tool direction by means of $AA_TOFF[i], no block is allowed to be activated in which the offset axis assignment i is modified (plane change, tool change cutting tool <=> turning tool, transformation change, TRAFOOF, TCARR=0, geometry axis change)

Reactions:
- Local alarm reaction.
- Alarm display.
- Interface signals are set.
- Correction block is reorganized.
- NC Stop on alarm at block end.

Remedy:
- Modify part program
- Program TOFFOF()

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

21700 Channel %1 block %3 axis %2 touch probe already deflected, edge polarity not possible

Parameters: %1 = Channel number

%2 = Axis name, spindle number

%3 = Block number

Definitions: The probe programmed under the keyword MEAS or MEAW is already deflected and has switched. For a further measuring operation, the probe signal must first be canceled (quiescent state of the probe).

The axis display is of no significance at the present time but an axis-specific evaluation has been planned for later stages of development.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Verify the start position of the measuring operation or check the probe signals. Are the cables and connectors in good order?

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

21701 Channel %1 block %3 axis %2 measurement not possible

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number

Definitions: Measurement level 2 (MEASA, MEAWA, MEAC).
There is an error in the programmed measurement task.
Possible causes:
- Invalid measurement mode
- Invalid probe
- Invalid encoder
- Invalid number of measurement signal edges
- Identical measurement signal edges are only programmable in mode 2
- Invalid FIFO number
- Mismatch between the number of FIFOs programmed and the number of probes used in the measurement task.

Further causes:
- A measurement task is already active (e.g. from a synchronized action).

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Correct the measurement tasks.

Program Continuation: Clear alarm with the RESET key. Restart part program

21702 Channel %1 block %3 axis %2 measurement aborted

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number

Definitions: The measurement block has ended (the programmed end position of the axis has been reached) but the activated touch probe has not yet responded.
Measurement level 2 (MEAWA, MEASA, MEAC)
Measured values cannot be converted to the workpiece coordinate system. The measured values of the GEO axes programmed in the measurement task are only available in the machine coordinate system.
Causes:
Not all GEO axes were programmed in the measurement task. At least one measured value is therefore missing for conversion back into the workpiece coordinate system.
Further causes:
The measurement tasks programmed for all GEO axis are not identical.

Reactions: - Alarm display.

Remedy: Verify the traversing movement in the measurements block.
- Is it necessary in all cases for the activated probe to have switched up to the specified axis position?
- Are the probe, cable, cable distributor, terminal connections in good order?

Either program all GEO axes explicitly or program the traversing movement with the POS[axis] command.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

21703 Channel %1 block %3 axis %2 touch probe not deflected, illegal edge polarity

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number

Definitions: The selected probe is not (!) deflected and therefore cannot record any measured value from the deflected to the non-deflected state.
Measurement level 2 (MEAWA, MEASA, MEAC)
The degree of deflection of the probe at the start of the measurement task is identical to the first programmed measurement signal edge. The test is only performed in mode 2.

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy:
- Check probe
- Check start positioning for measuring
- Check program

Program Continuation: Clear alarm with the RESET key. Restart part program

21740 Output value at analog output no. %1 has been limited

Parameters: %1 = No. of output

Definitions: The value range of the analog output n is limited by machine data 10330 FASTIO_ANA_OUTPUT_WEIGHT[n].

Reactions: - Alarm display.

Remedy: With $A_OUTA[..] = x no greater values can be programmed than permitted in the respective machine data.

Program Continuation: Clear alarm with the Delete key or NC START.

21750 Error during output of cam signals via timer

Definitions: The signal output activated by the MD 10480 SW_CAM_TIMER_FASTOUT_MASK via the hardware timer (independent of the clock grid) did not work. Cause: interpolation cycle is greater than 15 ms.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Shorten interpolation cycle (if at all possible).

Program Continuation: Switch control OFF - ON.

21760 Channel %1 block %2 too many auxiliary functions programmed

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The number of programmed auxiliary functions has exceeded the maximum permissible amount. This alarm can occur in conjunction with motion synchronous actions: The maximum number of auxiliary functions must not be exceeded in motion block and motion synchronous actions.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

21800 Channel %1 workpiece setpoint %2 reached

Parameters:
%1 = Channel number
%2 = Workpiece setpoint

Definitions: This alarm is activated via MD 27880 PART_COUNTER, bit 1:
The number of counted workpieces ($AC_ACTUAL_PARTS or $AC_SPECIAL_PARTS) is equal or already larger than the programmed value for the number of required workpieces ($AC_REQUIRED_PARTS).
At the same time, the channel VDI signal "Workpiece setpoint reached" is output.
The value for the number of the counted workpieces ($AC_ACTUAL_PARTS) is reset while the value of $AC_SPECIAL_PARTS remains.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.

Remedy: No program interrupt. Delete alarm display.

Program Continuation: Clear alarm with the Delete key or NC START.

22000 Channel %1 block %3 spindle %2 change of gear stage not possible

Parameters:
%1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: Automatic gear stage selection has been programmed with M40. The new M word is not in the present gear stage, but the spindle is not in "Open-loop control mode".
For automatic gear stage change (M40 in conjunction with spindle speed in address S) the spindle must be in "Open-loop control mode".

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Before the S word which requires a gear stage change, change into the open-loop control mode of the spindle.
Change to the open-loop control mode is carried out with:
- M03, M04, M05 or M41 ... M45 from axis mode and positioning mode
- Interface signal "Gear is changed" (DB 31 - 48, DBX 16.3) from oscillation mode

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

22010 Channel %1 block %3 spindle %2 actual gear stage differs from requested gear stage

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: The requested gear stage change has been concluded. The actual gear stage reported by the PLC as being engaged is not the same as the required gear stage called for by the NC. Note: Wherever possible, the requested gear stage should always be engaged.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Correct the PLC program.

Program Continuation: Clear alarm with the Delete key or NC START.

22011 Channel %1 block %3 spindle %2 change to programmed gear stage not possible

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: With the 'DryRun', 'ProgramTest' and 'SearchRunByProgTest' functions deselected, it is not possible in the Repos module to carry out a gear stage change to a previously programmed gear stage. This is the case, if the spindle is in the deselection block not active in speed control mode, as a slave axis or in a transformation. Execution of a gear stage change is avoided if the above mentioned functions are deselected by resetting bit 2 of machine data 35035 SPIND_FUNCTION_MASK.

Reactions: - Alarm display.

Remedy: Change deselection block or block search target block to speed control mode (M3, M4, M5, SBCOF). Set bit 2 of machine data 35035 SPIND_FUNCTION_MASK to 0.

Program Continuation: Clear alarm with the Delete key or NC START.

22012 Channel %1 block %2 leading spindle %3 is in simulation.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Leading spindle number

Definitions: When coupling, no synchronism can be achieved if the lead spindle/axis is in simulation mode and the following spindle/axis is not.

Reactions: - Alarm display.

Remedy: Set the following spindle/axis to simulation mode, or do not simulate the lead spindle/axis ($MA_CTRLOUT_TYPE). If the differing settings have been selected on purpose, the alarm can be suppressed with the machine data 11410 SUPPRESS_ALARM_MASK Bit21 = 1.

Program Continuation: Clear alarm with the Delete key or NC START.

22013 Channel %1 block %2 dependent spindle %3 is in simulation.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Following spindle number

Definitions: When coupling, no synchronism can be achieved if the following spindle/axis is in simulation mode and the lead spindle/axis is not.

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Set the lead spindle/axis to simulation mode, or do not simulate the following spindle/axis ($MA_CTRLOUT_TYPE). If the differing settings have been selected on purpose, the alarm can be suppressed with the machine data 11410 SUPPRESS_ALARM_MASK Bit21 = 1.

Program Continuation: Clear alarm with the Delete key or NC START.

22014 Channel %1 block %2. The dynamics of leading spindle %3 and dependent spindle %4 is too variably

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Leading spindle number
%4 = Following spindle number

Definitions: If the spindles / axes differ strongly in their dynamic behavior during coupling, synchronism cannot be achieved. The dynamics are dependent on many settings: Parameter block data, first of all line KV, balancing time, etc., feedforward control mode and feedforward setting parameter, FIPO mode, jerk filter and dynamic filter settings, DSC on/off. Among these are the following machine data: MA_VELO_FFW_WEIGHT, MA_FIPO_TYPE, VEL_FFW_TIME, MA_EQUIV_SPEEDCTRL_TIME, MA_POSCTRL_GAIN, AX_JERK_TIME, STIFFNESS_DELAY_TIME, PROFIBUS_ACTVAL_LEAD_TIME, PROFIBUS_OUTVAL_DELAY_TIME, CTRLOUT_LEAD_TIME

Reactions: - Alarm display.

Remedy: Use spindles/axes with the same dynamics. If the differing settings have been selected on purpose, the alarm can be suppressed with the machine data 11410 SUPPRESS_ALARM_MASK Bit21 = 1.

Program Continuation: Clear alarm with the Delete key or NC START.

22020 Channel %1 block %3 spindle %2 gear step change position not reached

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Block number, label

Definitions: Through the configuration of MA_GEAR_STEP_CHANGE_ENABLE[AXn] = 2, the spindle is traversed to the position stored in MA_GEAR_STEP_CHANGE_POSITION[AXn] before the actual gear step change. The required gear step change position has not been reached.

Reactions: - Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct sequence in the PLC.

Program Continuation: Clear alarm with the RESET key. Restart part program

22040 Channel %1 block %3 spindle %2 is not referenced with zero marker

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: The current position is not referenced with the MS position although reference is made to it.

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Correct NC part program. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.

Program Continuation: Clear alarm with the Delete key or NC START.

22045 Block %2 spindle/axis %3 not available in channel %1 because active in channel %4

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Axis name, spindle number

%4 = Number of the channel in which the spindle/axis is currently active.

Definitions: The specified spindle/axis is required in channel %1 for the correct execution of a function. The spindle/axis is currently active in the %4 channel. The constellation can only occur with replacement axes.

Problem case: A synchronized spindle coupling was programmed. The master spindle/axis is not contained in the channel programmed for the coupling at the time the coupling is activated. The master spindle/axis can be moved by FC18 or synchronized actions. When using FC18, please note that the master spindle/axis must be assigned to the channel which activates the coupling. When FC18 terminates, the master spindle/axis must not be assigned to another channel via PLC while the coupling is still active (VDI interface signals).

Reactions: - Alarm display.
- NC Stop on alarm.
- Interface signals are set.

Remedy:
- Program a GET for the master spindle/axis in the NC program before activating the coupling, or
- Assign the master spindle/axis to the channel that activated the coupling via PLC.

Program Continuation: Clear alarm with the Delete key or NC START.

22050 Channel %1 block %3 spindle %2 no transition from speed control mode to position control mode

Parameters: %1 = Channel number

%2 = Axis name, spindle number

%3 = Block number, label

Definitions:
- An oriented spindle stop (SPOS/SPOSA) has been programmed or the position control of the spindle was switched on with SPCON but no spindle encoder has been defined.
- When switching on the position control, the spindle speed is greater than the limiting speed of the measuring system.

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Spindle without attached encoder: Any NC language elements requiring the encoder signals must not be used.

Spindle with attached encoder: Enter the number of spindle encoders used in the MD NUM_ENCS.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

22051 Channel %1 block %3 spindle %2 reference mark not found

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: When referencing, the spindle turned through a greater distance than given in the axis-specific machine data 34060 REFP_MAX_MARKER_DIST, without receiving a reference mark signal. The check is performed for spindle positioning with SPOS or SPOSA when the spindle has not previously run with speed control (S=...).

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check and correct the machine data 34060 REFP_MAX_MARKER_DIST. The value entered states the distance in [mm] or [degrees] between 2 zero markers.

Program Continuation: Clear alarm with the RESET key. Restart part program

22052 Channel %1 block %3 spindle %2 no standstill on block change

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: The displayed spindle has been programmed as spindle or as axis even though a positioning operation is still running from the previous block (with SPOSA ... spindle positioning beyond block limits).

Example:

```
N100 SPOSA [2] = 100
 :
N125 S2 = 1000 M2 = 04 ; Error, if spindle S2 from block N100 is still running!
```

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Before programming the spindle/axis again using the SPOSA instruction, a WAITS command should be activated in order to wait for the programmed spindle position.

Example:

```
N100 SPOSA [2] = 100
 :
N125 WAITS (2)
N126 S2 = 1000 M2 = 04
```

Program Continuation: Clear alarm with the RESET key. Restart part program

22053 Channel %1 block %3 spindle %2 reference mode not supported

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: In the case of SPOS/SPOSA with an absolute encoder, only the referencing mode ENC_REFP_MODE = 2 is supported! SPOS/SPOSA does not support ENC_REFP_MODE = 6 at all!

© Siemens AG, 2002. All rights reserved

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Modify setting of ENC_REFP_MODE or change to JOG+REF and then reference.

Program Continuation: Clear alarm with the RESET key. Restart part program

22054 Channel %1 block %3 spindle %2 improper punching signal

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: If the punching signal is irregular between the punching strokes, this alarm is generated according to machine data.

Reactions: - Alarm display.

Remedy: Indicates poor condition of the punching hydraulics.

Program Continuation: Clear alarm with the Delete key or NC START.

22055 Channel %1 block %3 spindle %2 configured positioning speed is too high

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label

Definitions: The current position is not referenced with the MS position although reference is made to it.

Reactions: - Alarm display.

Remedy: Correct NC part program. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.

Program Continuation: Clear alarm with the Delete key or NC START.

22060 Channel %1 position control expected for axis/spindle %2

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The programmed coupling type (DV, AV) or the programmed function requires position control.

Reactions: - Alarm display.

Remedy: Activate position control, e.g. by programming SPCON.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

22062 Channel %1 axis %2 reference point approach: zero marker search velocity (MD) is not reached

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The configured zero marker search velocity is not reached.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Check active spindle speed limitations. Configure a lower zero marker search velocity $MA_REFP_VELO_SEARCH_MARKER. Check the tolerance range for the actual velocity $MA_SPIND_DES_VELO_TOL. Set a different referencing mode $MA_ENC_REFP_MODE != 7.

Program Continuation: Clear alarm with the RESET key. Restart part program

22064 Channel %1 axis %2 reference point approach: zero marker search velocity (MD) is too high

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The configured zero marker search velocity is too high. The encoder limit frequency is exceeded for the active measuring system.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Configure a lower zero marker search velocity $MA_REFP_VELO_SEARCH_MARKER. Check the encoder frequency configuration $MA_ENC_FREQ_LIMIT and $MA_ENC_FREQ_LIMIT_LOW. Set a different referencing mode ($MA_ENC_REFP_MODE != 7).

Program Continuation: Clear alarm with the RESET key. Restart part program

22065 Channel %1 tool management: Tool motion is not possible, as tool %2 with Duplo no. %3 is not in magazine %4

Parameters: %1 = Channel number
%2 = String (identifier)
%3 = Duplo no.
%4 = Magazine no.

Definitions: The desired tool motion command - triggered from the MMC or PLC - is not possible. The specified tool is not contained in the specified magazine. (NCK cannot contain tools that are not assigned to a magazine. With this kind of tool, no operations (motion, change) can be performed.)

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Check that the specified tool is contained in the desired magazine, or program another tool to be changed.

Program Continuation: Clear alarm with the Delete key or NC START.

22066 Channel %1 tool management: Tool change is not possible, as tool %2 with Duplo no. %3 is not in magazine %4

Parameters: %1 = Channel number
%2 = String (identifier)
%3 = Duplo no.
%4 = Magazine no.

Definitions: The desired tool change is not possible. The specified tool is not contained in the specified magazine. (NCK cannot contain tools that are not assigned to a magazine. With this kind of tool, no operations (motion, change) can be performed.)

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department.
- Check that the specified tool is contained in the desired magazine, or program another tool to be changed.
- Check whether the settings in machine data $MC_RESET_MODE_MASK, $MC_START_MODE_MASK and the associated machine data $MC_TOOL_RESET_NAME match the current definition data.

Program Continuation: Clear alarm with the RESET key. Restart part program

22067 Channel %1 tool management: tool change not possible since there is no tool available in tool group %2

Parameters: %1 = Channel number
%2 = String (identifier)

Definitions: The desired tool change is not possible. The specified tool group does not contain a tool which is ready for use and could be used for tool change. It is possible that all of the tools in question have been set to the 'Disabled' state by the tool monitoring function.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm at block end.

Remedy:
- Ensure that the specified tool group contains a tool that is ready for use when tool change is requested.
- This can be achieved, for example, by replacing disabled tools, or
- by releasing a disabled tool manually.
- Check whether the tool data are correctly defined. Have all intended tools in the group been defined with the specified identifier and loaded?

Program Continuation: Clear alarm with the RESET key. Restart part program

22068 Channel %1 block %2 tool management: no tool available in tool group %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String (identifier)

Definitions: The specified tool group does not contain a tool which is ready for use and could be used for tool change. It is possible that all of the tools in question have been set to the 'Disabled' state by the tool monitoring function. The alarm can occur for example in conjunction with the alarm 14710 (error on INIT block generation). In this specific situation, NCK attempts to replace the disabled tool located on the spindle with an available replacement tool (which does not exist in this error condition).

The user must resolve this conflict, for example, by removing the tool located on the spindle from the spindle by issuing a movement command (e.g. through MMC operation).

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Ensure that the specified tool group contains a tool that is ready for use when tool change is requested.
- This can be achieved, for example, by replacing disabled tools, or
- by releasing a disabled tool manually.
- Check whether the tool data are correctly defined. Have all intended tools in the group been defined with the specified identifier and loaded?

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key. Restart part program

22069 Channel %1 block %2 tool management: No tool available in tool group %3, program %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = String (identifier)
%4 = Program name

Definitions: The specified tool group does not contain a tool which is ready for use and could be used for tool change. It is possible that all of the tools in question have been set to the 'Disabled' state by the tool monitoring function. Parameter %4 = program name facilitates the identification of the program containing the programming command (tool selection) that caused the error. This can be a subprogram or cycle, etc., which can no longer be identified from the display. If the parameter is not specified, it is the currently displayed program.

Reactions: - Alarm display.
- Interface signals are set.
- Correction block is reorganized.

Remedy:
- Ensure that the specified tool group contains a tool that is ready for use when tool change is requested.
- This can be achieved, for example, by replacing disabled tools, or
- by releasing a disabled tool manually.
- Check whether the tool data are correctly defined. Have all intended tools in the group been defined with the specified identifier and loaded?

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

22070 TO unit %1 Please change tool T= %2 into magazine. Repeat data backup

Parameters: %1 = TO unit
%2 = T number of tool

Definitions: The alarm can only occur when the tool management function is active in the NCK. (TOOLMAN = tool management) A data backup of the tool/magazine data has been started. During the backup, the system detected that tools are still located in the buffer magazine (= spindle, gripper, ...). During the backup, these tools will lose the information which defines the magazine and location to which they are allocated.

It is therefore practical - assuming that the data are to be stored exactly as before - to ensure that all tools have been deposited in the magazine before the data backup!!

If this is not the case, some magazine locations will have the 'reserved' status when the data are loaded again. This 'reserved' status must then be reset manually.

For tools with fixed location coding, the loss of the information allocating their location in the magazine has the same effect as a general empty location search when they are returned to the magazine.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Ensure that no tools are located in the buffer magazine before the data backup. Repeat the data backup after removing the tools from the buffer magazine.

Program Continuation: Clear alarm with the Delete key or NC START.

22071 TO unit %1 tool %2 duplo no. %3 is active, but not in the magazine area under consideration

Parameters: %1 = TO unit
%2 = Tool identifier
%3 = Duplo number

© Siemens AG, 2002. All rights reserved

Definitions: The alarm can only occur when the tool management function is active in the NCK. Either the language command SETTA has been programmed or the corresponding operator action has been carried out via MMC, PLC, ... The alarm can also be triggered automatically by the NCK in the wear grouping function. It is detected that more than one tool from the tool group (tools with the same name/identifier) has the status "active".

The specified tool is either

from a non-considered magazine,

from a non-considered wear grouping,

or from a non-active wear grouping

in a buffer location (is neither magazine nor wear grouping).

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: The alarm is intended for information purposes. If only one tool in a group can be active at a time for technological reasons or for reasons of display, the "active" status must be canceled for the tool causing the error.

Otherwise, the alarm can be ignored or even suppressed via the machine data SUPPRESS_ALARM_MASK.

Typical reasons of display are present, if the operator works with the function 'definite D numbers', which can be displayed on Siemens MMC in a definite form only, if exactly one tool from a tool group has the status 'active'.

Before machining can be started or before the SETTA (or corresponding MMC operation, ...) language command is used, all tools of the magazine should have the status "not active".

One option to achieve this is programming SETTIA (or corresponding MMC operation, ...).

Program Continuation: Clear alarm with the Delete key or NC START.

22100 Channel %1 block %3 spindle %2 chuck speed exceeded

Parameters: %1 = Channel number

%2 = Axis name, spindle number

%3 = Block number, label

Definitions: The actual speed of the displayed spindle is greater than the value entered in the axis-specific machine data 35100 SPIND_VELO_LIMIT plus the tolerance value specified in machine data 35150 SPIND_DES_VELO_TOL.

If the drive actuator has been optimized properly, the alarm cannot occur in the 840D!

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Alarm display.
- Interface signals are set.
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check the setup and optimization data of the drive actuator in accordance with the Installation and Start-up Guide and make corrections.

Increase the tolerance window in machine data 35150 SPIND_DES_VELO_TOL.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

22200 Channel %1 spindle %2 axis stopped during tapping

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: When tapping with compensating chuck (G63) the drilling axis was stopped via the NC/PLC interface and the spindle continues to rotate. The thread and possibly also the tap were damaged as a result.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Provide an interlock in the PLC user program so that no axis stop can be initiated when tapping is active. If the tapping operation is to be terminated under critical machine conditions, the spindle and the axis should be stopped simultaneously if at all possible. Slight differences are then accommodated by the compensating chuck.

Program Continuation: Clear alarm with the RESET key. Restart part program

22250 Channel %1 spindle %2 axis stopped during thread cutting

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: The thread cutting axis has been stopped while a thread block was active.
The stop can be caused by VDI signals that cause the feed to be interrupted.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Please inform the authorized personnel/service department. Check the axis-specific/spindle-specific stop signals (DB 31 - 48, DBX 4.3).

Program Continuation: Clear alarm with the RESET key. Restart part program

22260 Channel %1 spindle %2 thread might be damaged

Parameters: %1 = Channel number
%2 = Axis name
%3 = Block number

Definitions: When DECODING SINGLE BLOCK has been selected and there is a chain of thread blocks, then machining pauses occur at the block limits until the next block is executed with the new NC Start.
In normal single block mode, the program is stopped by a higher-level logic only at the block boundaries at which no contour distortions or contour errors can occur. With chained thread blocks, this is the last thread block!

Reactions: - Alarm display.

Remedy: If only one thread block has been programmed, the alarm message can be ignored.
If there are several consecutive thread blocks, this machining section must not be executed in the automatic DECODING SINGLE BLOCK mode.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

22270 Channel %1 block %2 maximum velocity of thread axis at position %3 reached

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Position

© Siemens AG, 2002. All rights reserved

Definitions: The axis velocity is too high for thread cutting. The maximum feedrate was reached at the axis position indicated. The velocity of the thread axis depends on:
- The programmed thread pitch
- The programmed thread lead change (G34)
- The thread length (G34)
- The defined spindle speed (part program, FC18, synchronized action)
- The spindle override (path and individual axis overrides are ineffective)

Reactions: - Alarm display.
Remedy: Reduce the velocity for at least one of the above factors.
Program Continuation: Clear alarm with the Delete key or NC START.

22275 Channel %1 block %2 zero velocity of thread axis at position %3 reached

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Position
Definitions: An axis standstill was reached at the specified position during thread cutting with G35 due to the linear decrease in the thread lead. The standstill position of the thread axis depends on:
- Programmed thread lead decrease
- Thread length

Reactions: - Alarm display.
Remedy: Change at least one of the above factors.
Program Continuation: Clear alarm with the Delete key or NC START.

22280 Channel %1 in block %2: Prog. acceleration path too short %3, %4 required

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Prog. acceleration path
%4 = Required acceleration path
Definitions: In order to stay within the programmed acceleration path, the acceleration caused an overload on the thread axis. In order to accelerate the axis with the programmed dynamic response, the length of the acceleration path must be at least as large as the value in parameter %4.
The alarm is of the technological type and is output whenever bit 2 in $MN_ENABLE_ALARM_MASK is enabled. The MMC soft key 'Technology support' sets and clears this bit in the MD.
Reactions: - Alarm display.
Remedy: Modify part program or reset MD $MN_ENABLE_ALARMMASK bit 2.
Program Continuation: Clear alarm with the Delete key or NC START.

22320 Channel %1 block %2 PUTFTOCF command could not be transferred

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The cyclic transfer of the PUTFTOCF data block (fine tool compensation) could not be performed because the transfer area is already occupied.
Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Check the part program, in particular with regard to the other channels. Is a data block being transferred by another channel?
Program Continuation: Clear alarm with the RESET key. Restart part program

22321 Channel %1 axis %2 PRESET not allowed during traverse motion

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A preset command was given from MMC or PLC while an axis was traveling in JOG mode.
Reactions: - Alarm display.
- Interface signals are set.
Remedy: Wait until the axis is stationary.
Program Continuation: Clear alarm with the Delete key or NC START.

22322 Channel %1 axis %2 PRESET: illegal value

Parameters: %1 = Channel number
%2 = Axis name, spindle number
Definitions: The entered Preset value is too large (number format overflow).
Reactions: - Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- NC Start disable in this channel.
Remedy: Use more realistic (smaller) Preset values.
Program Continuation: Clear alarm with the RESET key. Restart part program

25000 Axis %1 hardware fault of active encoder

Parameters: %1 = Axis name, spindle number
Definitions: The signals of the currently active position actual value encoder (interface signal DB 31 - 48, DBX 1.5 = 1 or DBX 1.6 = 1) are missing, do not have the same phase, or exhibit grounding/short-circuit.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.
Remedy: Please inform the authorized personnel/service department. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found.
Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

25001 Axis %1 hardware fault of passive encoder

Parameters: %1 = Axis name, spindle number

Definitions: The signals from the position actual value encoder that is presently not active are missing, or they are not of the same phase, or they exhibit grounding/short-circuit.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found. Switch off the monitoring with the corresponding interface signal (DB 31 - 48, DBX 1.5 = 0 or DBX 1.6 = 0).

Program Continuation: Clear alarm with the RESET key. Restart part program

25010 Axis %1 pollution of measuring system

Parameters: %1 = Axis name, spindle number

Definitions: The encoder used for position control sends a contamination signal (only in measuring systems with contamination signal).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy: Please inform the authorized personnel/service department. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25011 Axis %1 pollution of passive encoder

Parameters: %1 = Axis name, spindle number

Definitions: The encoder not used for position control sends a contamination signal (only in measuring systems with contamination signal).

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.

Program Continuation: Clear alarm with the Delete key or NC START.

25020 Axis %1 zero mark monitoring of active encoder

Parameters: %1 = Axis name, spindle number

Definitions: The position encoder pulses between 2 zero marker pulses are counted (hardware function). A check is made in the interpolation cycle grid (standard setting 4 ms) as to whether the encoder always issues the same number of pulses between the zero markers. As soon as a difference is registered in the 4 counter bits of lowest significance, an alarm is triggered!

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions: - The NC switches to follow-up mode.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Axes of this channel must be re-referenced.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. The differences can result from transmission errors, disturbances, encoder hardware faults or from the evaluation electronics in the encoder used for position control. The actual value branch must therefore be checked:

1. Transmission path: Check the actual-value connector on the motor and on the FDD module for correct contacting, encoder cable for continuity, and also check for short-circuits or grounding (loose contact?).

2. Encoder pulses: Encoder power supply within the tolerance limits?

3. Evaluation electronics: Replace or reconfigure the drive module used.

Monitoring can be switched off by setting machine data 36310 ENC_ZERO_MONITORING [n]=... N ... encoder number: 1, 2) is set to 0.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25021 Axis %1 zero mark monitoring of passive encoder

Parameters: %1 = Axis name, spindle number

Definitions: Monitoring relates to the encoder that is not used by the position control! (Interface signal DB 31 - 48, DBX 1.5 = 0 or 1.6 = 0)

The position encoder pulses between 2 zero marker pulses are counted (hardware function). A check is made in the interpolation cycle grid (standard setting 4 ms) as to whether the encoder always issues the same number of pulses between the zero markers. As soon as a difference is registered in the 4 counter bits of lowest significance, an alarm is triggered!

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. The differences can result from transmission errors, disturbances, encoder hardware faults or from the evaluation electronics in the encoder used for position control. The actual value branch must therefore be checked:

1. Transmission path: Check the actual-value connector on the motor and on the FDD module for correct contacting, encoder cable for continuity, and also check for short-circuits or grounding (loose contact?).

2. Encoder pulses: Encoder power supply within the tolerance limits?

3. Evaluation electronics: Replace or reconfigure the drive module used.

Monitoring can be switched off by setting machine data ENC_ZERO_MON_ACTIVE [n]=... N ... encoder number: 1, 2) to 0.

Program Continuation: Clear alarm with the Delete key or NC START.

25022 Axis %1 encoder %2 warning %3

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

%3 = Error fine coding

© Siemens AG, 2002. All rights reserved

Definitions: This alarm occurs only with absolute encoders on the SIMODRIVE 611D, if zero mark monitoring has been activated for them (cf. $MA_ENC_ZERO_MONITORING): In this case, the absolute position of the absolute encoder could not be read without any errors:
Breakdown of error fine codings:
(Bit 0 not used)
Bit 1 Parity error
Bit 2 Alarm bit of the encoder
Bit 3 CRC error
Bit 4 Timeout start bit for EnDat transfer is missing
Only display of this alarm, since the absolute position itself is not required at this time for control/contour.
Frequent occurring of this alarm indicates that absolute encoder transfer or the absolute encoder itself are faulty and that the absolute value determined with the next encoder selection or Power On situation could possibly be wrong.

Reactions: - Alarm display.

Remedy: Replace the encoder, replace or screen the encoder cable (or deactivate zero mark monitoring).

Program Continuation: Clear alarm with the Delete key or NC START.

25030 Axis %1 actual velocity alarm limit

Parameters: %1 = Axis name, spindle number

Definitions: If the axis has at least one active encoder, then the actual speed of the axis is cyclically checked in the IPO cycle. If there are no errors, the actual velocity can never become greater than specified in the axis-specific MD 36200 AX_VELO_LIMIT (threshold for velocity monitoring). This threshold value in [mm/min, rev/min] is input by an amount that is about 5 to 10% greater than that which can occur at maximum traversing velocity. Drive errors can result in the velocity being exceeded and the alarm is then triggered.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the speed setpoint cable (bus cable). Check the actual values and direction of position control. Change position control direction if the axis rotates uncontrollably -> axis-specific MD 32110 ENC_FEEDBACK_POL [n] = < -1, 0, 1 >. Increase the monitoring limit value in MD 36200 AX_VELO_LIMIT.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25031 Axis %1 actual velocity warning limit

Parameters: %1 = Axis name, spindle number

Definitions: The present velocity actual value is more than 80% of the limit value defined in the machine data -- not used --

Reactions: - Alarm display.

Remedy: -

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

25040 Axis %1 standstill monitoring

Parameters: %1 = Axis name, spindle number

Definitions: The NC monitors to ensure that the position is held at zero speed. Monitoring is started after a time that can be set for a specific axis in the machine data 36040 STSTILL_DELAY_TIME after interpolation has ended. A constant check is made to determine whether the axis remains within the tolerance range given in MD 36030 STSTILL_POS_TOL.

The following cases are possible:

1. The interface signal SERVO ENABLE (DB31 - 48, DBX 2.1) is zero because the axis has jammed mechanically. Due to mechanical influences (e.g. high machining pressure), the axis is pushed away from the permissible position tolerance.

2. With closed position control loop (without jamming) - interface signal SERVO ENABLE (DB 31 - 48, DBX 2.1) is 1 - the axis is pushed away from its position by mechanical forces with a small gain in the position control loop.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- The NC switches to follow-up mode.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department.
- Check MD 36040 STSTILL_DELAY_TIME and MD 36030 STSTILL_POS_TOL; increase if necessary. The value must be greater than the machine data "Exact stop - coarse" ($MA_STOP_LIMIT_COARSE).
- Estimate machining forces and reduce if necessary by setting a lower feed or a higher rotational speed.
- Increase clamping pressure.
- Increase the gain in the position control loop by improved optimization (Kv factor MD 32200 POSCTRL_GAIN, 611D drive).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25042 Axis %1 standstill monitoring during torque/force limitation

Parameters: %1 = Axis name, spindle number

Definitions: The defined end position was not reached within the time specified in the machine data.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- If the drive torque (FXST) was set too low with the result that the force of the motor was not sufficient to reach the end position -> increase FXST.
- If the machined part is slowly deformed, there may be a delay in reaching the end position -> increase MD 36042 FOC_STANDSTILL_DELAY_TIME.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

25050 Axis %1 contour monitoring

Parameters: %1 = Axis name, spindle number

Definitions: The NCK calculates for each interpolation point (setpoint) of an axis the actual value that should result based on an internal model. If this calculated actual value and the true machine actual value differ by a larger amount than given in the machine data 36400 CONTOUR_TOL, then the program is aborted and the alarm message is issued.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Check whether the tolerance value set in MD 36400: CONTOUR_TOL is too small.
- Check optimization of the position controller (Kv factor in the machine data 32200 POSCTRL_GAIN) to establish whether the axis follows the given setpoint without overshooting. Otherwise, the speed controller optimization must be improved or the Kv servo gain factor must be reduced.
- Improvement of speed controller optimization
- Check the mechanics (smooth running, inertial masses).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25060 Axis %1 speed setpoint limitation

Parameters: %1 = Axis name, spindle number

Definitions: The speed setpoint has exceeded its upper limit for a longer period than allowed.

The maximum speed setpoint is limited to a certain percentage with the axis-specific machine data 36210 CTRLOUT_LIMIT. The input value of 100% corresponds to the rated speed of the motor and hence the rapid traverse velocity (default values: 840D=110%, FM-NC=100%).

If the values are exceeded for a short time, then this is tolerated provided they do not last longer than allowed for in the axis-specific MD 36220 CTRLOUT_LIMIT_TIME. The setpoint is limited during this time to the maximum value that has been set (MD 36210).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. If the drive controller has been set correctly and if the machining conditions are those that normally prevail, then this alarm should not occur.

© Siemens AG, 2002. All rights reserved

- Check actual values: Local tight running of the carriage, speed dip by torque reduction due to contact with workpiece/tool, travel against fixed obstacle, etc.
- Check direction of position control: Does the axis continue to rotate without control (not on 611D drives)?
- Check the speed setpoint cable.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25070 Axis %1 drift value too large

Parameters: %1 = Axis name, spindle number

Definitions: Only in FM-NC with analog drives!

The permissible maximum value of drift (internal, integrated drift value of automatic drift compensation) has been exceeded during the last compensation operation. The permissible maximum value is defined in the axis-specific machine data 36710 DRIFT_LIMIT. The drift value itself is not limited.

Automatic drift compensation: MD 36700 DRIFT_ENABLE=1

The difference between actual and setpoint position (drift) is checked cyclically in the IPO cycle when the axes are at zero speed. The difference is compensated automatically to zero by slowly integrating an internal drift value.

Drift compensation by hand: MD 36700 DRIFT_ENABLE=0

A static offset can be added to the speed setpoint in the machine data 36720 DRIFT_VALUE. This is not included in the drift monitoring because it acts like a voltage zero offset.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Adjust the drift compensation with the automatic drift compensation switched off at the drive until the position lag is approximately zero. Then reactivate the automatic drift compensation in order to balance out the dynamic drift changes (effects of heating up).

Program Continuation: Clear alarm with the Delete key or NC START.

25080 Axis %1 positioning monitoring

Parameters: %1 = Axis name, spindle number

Definitions: For blocks in which "exact stop" is effective, the axis must have reached the exact stop window after the positioning time given in the axis-specific MD 36020 POSITIONING_TIME.

Exact stop coarse: MD 36000 STOP_LIMIT_COARSE

Exact stop fine: MD 36010 STOP_LIMIT_FINE

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Check whether the exact stop limits (course and fine) correspond to the dynamic possibilities of the axis, otherwise increase them, if necessary in connection with the positioning time set in MD 36020 POSITIONING_TIME. Check speed controller/position controller optimization; select highest possible gains. Check setting of Kv factor (MD 32200 POSCTRL_GAIN) and increase if necessary.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25100 Axis %1 measuring system switchover not possible

Parameters: %1 = Axis name, spindle number

Definitions: The prerequisites are not satisfied for the required encoder switchover:

1. The newly selected encoder must be in the active state (DB 31 - 48, DBX 1.5 or 1.6 = 1 "Position measuring system 1/2")

2. The actual value difference between the two encoders is greater than the value in the axis-specific MD 36500 ENC_CHANGE_TOL ("Maximum tolerance for position actual value switchover").

Activation of the measuring system concerned takes place in accordance with the interface signals: "Position measuring system 1" (DB 31 - 48, DBX 1.5) and "Position measuring system 2" (DB 31 - 48, DBX 1.6), i.e. the position control is now operated with this measuring system. The other measuring system is switched over to follow-up mode. If both interface signals are set to "1", then only the 1st measuring system is active; if both interface signals are set to "0", the axis is parked.

Changeover takes place as soon as the interface signals have changed, even if the axis is in motion!

Reactions:
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- NC Stop on alarm.
- NC Start disable in this channel.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. When referencing the active position actual value encoder, the actual value system of the inactive encoder is set to the same reference point value as soon as phase 3 has been concluded. A later positional difference between the 2 actual value systems can have occurred only as the result of an encoded defect or a mechanical displacement between the encoders.

- Check the encoder signals, actual value cable, connectors.
- Check the mechanical fastenings (displacement of the measuring head, mechanical twisting possible).
- Increase the axis-specific MD 36500 ENC_CHANGE_TOL.

Program continuation is not possible. The program must be aborted with "Reset", then program execution can be reinitiated with NC Start, if necessary at the interruption point after "Block search with/without calculation".

Program Continuation: Clear alarm with the RESET key. Restart part program

25105 Axis %1 measuring systems differ considerably

Parameters: %1 = Axis name, spindle number

Definitions: The two measuring systems differ considerably, i.e. the cyclically monitored actual value difference between the two measuring systems is greater than the associated tolerance value set in the machine data $MA_ENC_DIFF_TOL. This can only occur when both measuring systems are active ($MA_NUM_ENCS = 2) and referenced. The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check machine data for the active, selected encoders. Check the machine data relating to encoder ($MA_ENC_DIFF_TOL) tolerance.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25110 Axis %1 selected encoder not available

Parameters: %1 = Axis name, spindle number

Definitions: The selected encoder does not correspond to the maximum number of encoders in the axis-specific machine data 30200 NUM_ENCS, i.e. the 2nd encoder does not exist.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Enter the number of actual value encoders used for this axis in the machine data 30200 NUM_ENCS ("Number of encoders").

Input value 0: Axis without encoder -> e.g. spindle

Input value 1: Axis with encoder -> default setting

Input value 2: Axis with 2 encoders -> e.g. direct and indirect measuring system

Program Continuation: Clear alarm with the Delete key or NC START.

25200 Axis %1 requested set of parameters invalid

Parameters: %1 = Axis name, spindle number

Definitions: A new parameter set has been requested for the positioning control. The number of this parameter set is beyond the permissible limit (8 parameter sets: 0 ... 7 available).

Reactions: - NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Check the axis-specific/spindle-specific interface signals (DB 31 - 48, DBX 9.0, 9.1 and 9.2 "Select parameter set servo A, B, C").

One parameter set includes the following machine data:

- Modify MD 31050: DRIVE_AX_RATIO_DENOM [n]
- Modify MD 31060: DRIVE_AX_RATIO_NUMERA [n]
- Modify MD 32200: POSCTRL_GAIN [n]
- Modify MD 32800: EQUIV_CURRCTRL_TIME [n]
- Modify MD 32810: EQUIV_SPEEDCTRL_TIME [n]
- Modify MD 32910: DYN_MATCH_TIME [n]
- Modify MD 36200: AX_VELO_LIMIT [n]

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

25201 Axis %1 drive fault

Parameters: %1 = Axis name, spindle number

Definitions: The drive signals a serious fault of status class 1 (ZK1). The exact cause of the fault can be recognized by evaluating the following drive alarms which are output in addition:

Alarm 300 500, alarms 300 502 - 300 505, alarm 300 508, alarm 300 515, alarm 300 608, alarm 300 612, alarm 300 614, alarms 300 701 - 300 761, alarm 300 799.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Evaluation of the drive alarms listed above.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25202 Axis %1 waiting for drive

Parameters: %1 = Axis name, spindle number

Definitions: Drive group error (self-clearing).

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Wait for the drive. 25202 reveals similar problems to alarm 25201 (see this alarm). The alarm is active continuously during power-up if the drive does not communicate (e.g. Profibus connector removed). Otherwise, the alarm is active only briefly and is replaced by alarm 25201 after an internal timeout in the event of a permanent problem.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

26000 Axis %1 clamping monitoring

Parameters: %1 = Axis name, spindle number

Definitions: The clamped axis is to be pushed out of its setpoint position. The permissible difference is defined in the axis-specific machine data 36050 CLAMP_POS_TOL.

Clamping of an axis is activated with the axis-specific interface signal DB 31 - 48, DBX 2.3: "Clamping process active".

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Determine the position deviation to the setpoint position and, depending on the results, either increase the permissible tolerance in the MD or mechanically improve the clamping (e.g. increase clamping pressure).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26001 Axis %1 parameterization error: friction compensation

Parameters: %1 = Axis name, spindle number

Definitions: The parameterization of the adaptation characteristic in the quadrant error compensation is not allowed because acceleration value 2 (MD 32560 FRICT_COMP_ACCEL2 is not between acceleration value 1 (MD 32550 FRICT_COMP_ACCEL1) and acceleration value 3 (MD 32570 FRICT_COMP_ACCEL3).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the setting parameters of the quadrant error compensation (friction compensation), if necessary switch off the compensation with MD 32500 FRICT_COMP_ENABLE.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26002 Axis %1 encoder %2 parameterization error: number of encoder marks

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

Definitions: 1. Rotary measuring system ($MA_ENC_IS_LINEAR[]==FALSE)

The number of encoder marks set in MD31020 $MA_ENC_RESOL[] does not correspond to the value in the drive machine data MD1005 or zero has been entered in one of the two machine data.

2. Absolute measuring system with EnDat interface ($MA_ENC_TYPE[]==4)

On absolute encoders, the resolution of the incremental and absolute track supplied by the drive is also checked for consistency.

- Motor measuring system: MD1005, MD1022
- Direct measuring system: MD1007, MD1032

The two drive machine data must have a defined relation to one another. If the conditions listed below are not fulfilled, an alarm is output.

2.1 Rotary measuring system ($MA_ENC_IS_LINEAR[] == FALSE)

MD1022/MD1005 == 4 * n [n=1,2,3...] (motor measuring system)

MD1032/MD1007 == 4 * n [n=1,2,3...] (direct measuring system)

2.2 Linear measuring system ($MA_ENC_IS_LINEAR[] == TRUE)

MD1005/MD1022 == 4 * n [n=1,2,3...] (motor measuring system)

MD1007/MD1032 == 4 * n [n=1,2,3...] (direct measuring system)

© Siemens AG, 2002. All rights reserved

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Adjust machine data. For absolute encoders, pending drive alarms indicating encoder problems should be evaluated, if necessary.
They could be the cause of incorrect entries in MD1022/MD1032 which are read out of the encoder by the drive.

Program Continuation: Switch control OFF - ON.

26003 Axis %1 parameterization error: lead screw pitch

Parameters: %1 = Axis name, spindle number

Definitions: The pitch of the ballscrew/trapezoidal leadscrew set in the axis-specific machine data 31030 LEADSCREW_PITCH is zero.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Determine the leadscrew pitch (specify the machine manufacturer or pitch measurement with spindle cover removed) and enter it in the machine data 31030: LEADSCREW_PITCH (mostly 10 or 5 mm/rev.).

Program Continuation: Switch control OFF - ON.

26004 Axis %1 encoder %2 parameterization error: grid point distance with linear encoders

Parameters: %1 = Axis name, spindle number
%2 = Encoder number

Definitions: The encoder grid point distance set in the axis-specific MD 31010 ENC_GRID_POINT_DIST is zero.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enter the encoder grid point distance according to the data given by the machine (or measuring device) manufacturer in the machine data 31010 ENC_GRID_POINT_DIST.

Program Continuation: Switch control OFF - ON.

26005 Axis %1 parameterization error: output rating

Parameters: %1 = Axis name, spindle number

Definitions: The output evaluation of the analog speed setpoint set in the machine data 32250 RATED_OUTVAL or in MD 32260 RATED_VELO is zero.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The nominal output voltage in [%] of the maximum setpoint value (10 V) is entered in the machine data 32250 RATED_OUTVAL, at which the rated motor speed in [degrees/s] is to be reached (machine data 32260 RATED_VELO).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26006 Axis %1 encoder %2 encoder type/output type %3 not possible

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

%3 = Encoder type/output type

Definitions: Not every encoder type or output type is suitable for both the FM-NC and the 840D.

Permissible settings for 840D:

MD 30240 ENC_TYPE

= 0 Simulation

= 1 Signal generator

= 2 Square-wave encoder

MD 30130 CTRLOUT_TYPE

= 0 Simulation

= 1 Standard

© Siemens AG, 2002. All rights reserved

Permissible settings for FM-NC:
MD 30240 ENC_TYPE
= 0 Simulation
= 3 Step motor control
= 4 FM module position
MD 30130 CTRLOUT_TYPE
= 2 Step motor control
= 3 FM module position
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check machine data MD 20240 ENC_TYPE and/or MD 30130 CTRLOUT_TYPE and make the necessary corrections.

Program Continuation: Switch control OFF - ON.

26007 Axis %1 QEC: invalid coarse step size

Parameters: %1 = Axis name, spindle number

Definitions: The course step width for QEC must be within the range 1 <= course step width <= maximum value of MD 18342 MM_QEC_MAX_POINTS (currently 1025), because a greater number of values would exceed the available memory space.

Reactions: - Alarm display.

Remedy: Modify the system variable $AA_QEC_COARSE_STEPS accordingly.

Program Continuation: Clear alarm with the RESET key. Restart part program

26008 Axis %1 QEC: invalid fine step size

Parameters: %1 = Axis name, spindle number

Definitions: The fine step size for quadrant error compensation must be in the range 1 <= fine step size <= 16 because this value has an influence on the computation time of the QEC.

Reactions: - Alarm display.

Remedy: Modify the system variable $AA_QEC_FINE_STEPS accordingly.

Program Continuation: Clear alarm with the RESET key. Restart part program

26009 Axis %1 QEC: memory overflow

Parameters: %1 = Axis name, spindle number

Definitions: The product of the data $AA_QEC_COARSE_STEPS+1 and $AA_QEC_FINE_STEPS must not exceed the maximum number of the characteristic curve points (MD 18342 MM_QEC_MAX_POINTS). With a direction-dependent characteristic, this criterion applies to 2 * ($AA_QEC_COARSE_STEPS+1) * $AA_QEC_FINE_STEPS!

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Either increase 18342 MM_QEC_MAX_POINTS or reduce $AA_QEC_COARSE_STEPS and/or $AA_QEC_FINE_STEPS.

Program Continuation: Clear alarm with the RESET key. Restart part program

26010 Axis %1 QEC: invalid acceleration characteristic

Parameters: %1 = Axis name, spindle number

Definitions: $AA_QEC_ACCEL_1/2/3: The acceleration characteristic is divided into three areas. In each area there is a different quantization of the acceleration steps. The defaults should be changed only if compensation is inadequate in these acceleration areas.

The defaults are as follows:

- $AA_QEC_ACCEL_1 with approx. 2% of maximum acceleration ($AA_QEC_ACCEL_3),
- $AA_QEC_ACCEL_2 with approx. 60% of maximum acceleration ($AA_QEC_ACCEL_3),
- $AA_QEC_ACCEL_3 with maximum acceleration (32300 MAX_AX_ACCEL).

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Enter the values correctly: 0 < $AA_QEC_ACCEL_1 < $AA_QEC_ACCEL_2 < $AA_QEC_ACCEL_3.

Program Continuation: Clear alarm with the RESET key. Restart part program

26011 Axis %1 QEC: invalid measuring periods

Parameters: %1 = Axis name, spindle number

Definitions: $AA_QEC_MEAS_TIME_1/2/3: measuring time to determine the error criterion.

The measuring period begins when the criterion for activating the compensation value has been satisfied (the desired velocity changes the sign). The end is defined by the machine data values. In general, different measuring times are required for the three characteristic ranges. The presettings should be changed only if a problem occurs. The three data apply in each case for the three corresponding acceleration ranges.

1. $AA_QEC_MEAS_TIME_1 specifies the measuring time (for determining the error criterion) for accelerations in the range between 0 and $AA_QEC_ACCEL_1.

2. $AA_QEC_MEAS_TIME_2 specifies the measuring time in the range from $AA_QEC_ACCEL_1 to $AA_QEC_ACCEL_2.

3. $AA_QEC_MEAS_TIME_3 specifies the measuring time in the range from $AA_QEC_ACCEL_2 to $AA_QEC_ACCEL_3 and beyond.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Enter the values correctly: 0 < $AA_QEC_MEAS_TIME_1 < $AA_QEC_MEAS_TIME_2 < $AA_QEC_MEAS_TIME_3.

Program Continuation: Clear alarm with the RESET key. Restart part program

26012 Axis %1 QEC: feed forward control not active

Parameters: %1 = Axis name, spindle number

Definitions: The error criterion for determining the quadrant error necessitates a correctly set feedforward control.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- Alarm display.

Remedy: Switch on feedforward control and set it correctly.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26014 Axis %1 machine data %2 invalid value

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier

Definitions: Machine data includes a value that is not valid.

Reactions: - NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Repeat entry with correct value and then Power On.

Program Continuation: Switch control OFF - ON.

26015 Axis %1 machine data %2[%3] invalid value

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier
%3 = Index: MD array index

Definitions: Machine data includes a value that is not valid.

Reactions: - NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Repeat entry with correct value and then Power On.

Program Continuation: Switch control OFF - ON.

26016 Axis %1 machine data %2 invalid value

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier

Definitions: Machine data includes a value that is not valid.

Reactions: - NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Repeat entry with correct value and then Reset.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

26017 Axis %1 machine data %2[%3] invalid value

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier
%3 = Index: MD array index

Definitions: Machine data includes a value that is not valid.

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Repeat entry with correct value and then Reset.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26018 Axis %1 setpoint output drive %2 used more than once

Parameters: %1 = Axis name, spindle number
%2 = Drive number

Definitions: A setpoint has been selected more than once. The machine data 30110 $MA_CTRLOUT_MODULE_NR contains the same value for different axes.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Avoid dual assignment of the setpoint by correcting 30110 $MA_CTRLOUT_MODULE_NR. Also check the selected bus type $MA_CTROUT_SEGMENT_NR.

Program Continuation: Switch control OFF - ON.

26019 Axis %1 encoder %2 measurement not possible with this controller module

Parameters: %1 = NC axis number
%2 = Encoder number

Definitions: If the MD $MN_DRIVE_DIAGNOSIS[8] contains a value not equal to zero, then the control has found at least one control module which does not support measuring. Measuring was programmed from the part program for the associated axis.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: If possible, modify the measuring motion such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis. Otherwise, exchange the controller module for one that supports measuring. See MD $MN_DRIVE_DIAGNOSIS[8].

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

26020 Axis %1 encoder %2 hardware fault %3 during encoder initialization

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

%3 = Error fine coding

Definitions: Error during initialization of encoder (refer to additional information for absolute encoder interface from error fine coding).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- The NC switches to follow-up mode.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Axes of this channel must be re-referenced.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)

Remedy: Please inform the authorized personnel/service department. Rectify hardware error, replace encoder if necessary. Make sure that an appropriate control module supporting this function is available with EnDat or SSI absolute encoders.

Bit nos. and their significance:

Bit 0: Lighting failed

Bit 1: Signal amplitude too small

Bit 2: Position value incorrect

Bit 3: Overvoltage

Bit 4: Undervoltage

Bit 5: Overcurrent

Bit 6: Battery change necessary

Bit 7: Control check error, Note: SW 4.2 and higher, synchronous linear motor

Bit 8: EnDat encoder, incorrect overlapping, Note: SW 4.2 and higher, synchronous linear motor

Bit 9: C/D track error on encoder ERN1387 or EQN encoder connected or incorrectly configured (not on EQN, MD 1011)

Bit 10: Log cannot be aborted or old hardware

Bit 11: SSI level detected on data line or no encoder connected or incorrect encoder cable (ERN instead of EQN)

Bit 12: Timeout while reading measuring value

Bit 13: CRC error

Bit 14: Wrong IPU submodule for direct measuring signal, Note: Only with 611D expansion

Bit 15: Encoder faulty

Program Continuation: Switch control OFF - ON.

26022 Axis %1 encoder %2 measurement with simulated encoder not possible

Parameters: %1 = NC axis number

%2 = Encoder number

Definitions: Alarm occurs on the control when a measurement was made without the encoder hardware (simulated encoder).

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy:
- Please inform the authorized personnel/service department. If possible, modify the measuring motion such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis.
- Ensure that measurement is not taking place with simulated encoders (MD $MA_ENC_TYPE).

Program Continuation: Clear alarm with the RESET key. Restart part program

26024 Axis %1 machine data %2 value changed

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier

Definitions: Machine data contains an invalid value and therefore has been changed by the software.

Reactions: - Alarm display.

Remedy: Check MD.

Program Continuation: Clear alarm with the RESET key. Restart part program

26025 Axis %1 machine data %2[%3] value changed

Parameters: %1 = Axis name, spindle number
%2 = String: MD identifier
%3 = Index: MD array index

Definitions: Machine data contains an invalid value and therefore has been changed by the software internally to a valid value.

Reactions: - Alarm display.

Remedy: Check MD.

Program Continuation: Clear alarm with the RESET key. Restart part program

26030 Axis %1 encoder %2 absolute position lost

Parameters: %1 = Axis name, spindle number
%2 = Encoder number

Definitions: The absolute position of the absolute encoder has become invalid because
- on changing parameter block a changed gear stage ratio was identified between encoder and processing or
- the encoder has been replaced (the absolute encoders serial number has changed).

Reactions: - Mode group not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Rereferencing/resynchronization of the absolute encoder; attach encoder on the load side and configure correctly (e.g. MD 31040 ENC_IS_DIRECT).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26031 Axis %1 configuration error master-slave

Parameters: %1 = Axis name, spindle number

Definitions: The alarm is output when the same machine axis has been configured as a master and a slave axis. Each of the axes in the master/slave link can be operated either as master or slave.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Check machine data for all linked axes and correct if necessary:
- MD 37250 $MA_MS_ASSIGN_MASTER_SPEED_CMD
- MD 37252 $MA_MS_ASSIGN_MASTER_TORQUE_CTR.

Program Continuation: Clear alarm with the RESET key. Restart part program

26032 Axis %1 master-slave not configured

Parameters: %1 = Axis name, spindle number

Definitions: The master-slave coupling could not be activated because of incomplete configuration.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Check the current configuration of the master-slave coupling.

The configuration can be modified via the MASLDEF instruction or the machine data MD37250 $MA_MS_ASSIGN_MASTER_SPEED_CMD and MD37252 $MA_MS_ASSIGN_MASTER_TORQUE_CTR.

Program Continuation: Clear alarm with the RESET key. Restart part program

26050 Axis %1 parameter set change from %2 to %3 not possible

Parameters: %1 = Axis name, spindle number

%2 = Index: current parameter block

%3 = Index: new parameter block

Definitions: The parameter block change cannot be performed without jumps. This is due to the content of the parameter block to be switched on, e.g. different load gear factors.

Reactions:
- Alarm display.
- Interface signals are set.
- Local alarm reaction.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- NC Start disable in this channel.

Remedy: In following cases, the parameter block change is carried out via MD 31060 and MD 31050 without an alarm, even with different load gear ratio settings:

© Siemens AG, 2002. All rights reserved

1. In speed-controlled and follow-up mode.
2. With position control with the direct encoder.
3. With position control with the indirect encoder only within the position window (MD 36500 > actual position > MD 36500).

Program Continuation: Clear alarm with the RESET key. Restart part program

26051 Channel %1 in block %2 unanticipated stop crossed in continuous path mode

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The path interpolation did not stop, as required, at the end of the block, but will only decelerate to a standstill in the next block. This error situation occurs if the stop at block change was not planned by the path interpolation or was not detected early enough. A possible cause is that the PLC changed the spindle speed when $MA_SPIND_ON_SPEED_AT_IPO_START > 0, and the machine has to wait until the spindle has returned to the setpoint range. Another possible cause is that a synchronized action needs to be finished before the path interpolation continues. The alarm is only output if $MN_TRACE_SELECT = 'H400'. The alarm output is normally suppressed. - $MN_TRACE_SELECT has SIEMENS password protection.

Reactions: - Alarm display.

Remedy: $MA_SPIND_ON_SPEED_AT_IPO_START = 1. Program G09 before the alarm output in the block to allow the path interpolation to stop as planned.

Program Continuation: Clear alarm with the Delete key or NC START.

26052 Channel %1 in block %2: path velocity too high for auxiliary function output

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: This alarm usually occurs in a block with auxiliary function output during a movement. In this case, the wait for acknowledgement of the auxiliary function was longer than planned.
The alarm occurs if internal control inconsistencies cause continuous path mode (G64, G641, ...) to be blocked unexpectedly.
The path interpolation stops abruptly at the end of the block indicated in the message (regenerative stop). On the next block change, the path continues unless the abrupt stop has caused an error in the position controller (e.g. because the $MA_CONTOUR_TOL setting was over-sensitive).

Reactions: - Alarm display.

Remedy:
- If the alarm occurred in a block with auxiliary function output during the movement: from SW 5.1 or higher, increase machine $MN_PLC_CYCLE_TIME_AVERAGE or
- program G09 in the block indicated in the message to allow the path interpolation to stop as planned.

Program Continuation: Clear alarm with the Delete key or NC START.

26070 Channel %1 axis %2 cannot be controlled by the PLC, max. number exceeded

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: An attempt has been made to control more axes than allowed from the PLC.

Reactions: - Interface signals are set.
- Alarm display.

Remedy: Check the machine data MD_NUM_MAX_PLC_CNTRL_AXES and correct if necessary or reduce the number of PLC-controlled axes.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

26072 Channel %1 axis %2 cannot be controlled by the PLC

Parameters: %1 = Channel number
%2 = Axis name, spindle number

Definitions: Axis cannot be made a PLC-controlled axis. For the time being, the axis cannot be controlled at any state from the PLC.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Use Release or Waitp to make the axis a neutral one.

Program Continuation: Clear alarm with the Delete key or NC START.

26074 Channel %1 switching off PLC control of axis %2 not allowed in the current state

Parameters: %1 = Channel
%2 = Axis, spindle

Definitions: The PLC can return the control rights for an axis to program execution only if the axis is in the READY state.

Reactions: - Alarm display.
- Interface signals are set.
- Interpreter stop
- NC Start disable in this channel.

Remedy: Activate axial RESET and repeat procedure.

Program Continuation: Clear alarm with the Delete key or NC START.

26080 Channel %1 retraction position of axis %2 not programmed or invalid

Parameters: %1 = Channel
%2 = Axis, spindle

Definitions: No retraction position has been programmed for the axis trigger time or the position became invalid.

Reactions: - Alarm display.

Remedy: Preset value by means of POLFA(Axis,Type,Pos), with type = 1 (absolute) or type = 2 (incremental); type = 0 specifies the position as invalid.

Program Continuation: Clear alarm with the Delete key or NC START.

26081 Channel %1 axis trigger of axis %2 was activated, but axis is not PLC-controlled

Parameters: %1 = Channel
%2 = Axis, spindle

Definitions: The axis trigger for single axis was initiated. However, the axis is not PLC-controlled at the trigger time (therefore no single axis) or the position became invalid.

Reactions: - Alarm display.

Remedy: Preset axis PLC-controlled (declare single axis).

Program Continuation: Clear alarm with the Delete key or NC START.

26100 Axis %1 drive %2 sign of life missing

Parameters: %1 = Axis name, spindle number
%2 = Drive number

Definitions: The sign-of-life cell is no longer being updated by the drive.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Restart drive, check drive software.

Program Continuation: Switch control OFF - ON.

26101 Axis %1 drive %2 communication failure

Parameters: %1 = Axis name, spindle number
%2 = Drive number

Definitions: The drive is not communicating.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy:
- Check the bus configuration.
- Check the interface (connector removed, option module inactive, etc.).

Program Continuation: Clear alarm with the RESET key. Restart part program

26102 Axis %1 drive %2 sign of life missing

Parameters: %1 = Axis name, spindle number
%2 = Drive number

Definitions: The sign-of-life cell is no longer being updated by the drive.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy:
- Check the cycle settings.
- Increase the cycle time if necessary.
- Power-up the drive again.
- Check drive software.

Program Continuation: Clear alarm with the RESET key. Restart part program

26105 Drive of axis %1 not found

Parameters: %1 = Axis name, spindle number

Definitions: The drive configured for the specified axis could not be found. For example, a Profibus slave was configured on the NC but is not contained in SDB1000.

© Siemens AG, 2002. All rights reserved

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Possible causes:
- $MA_CTRLOUT_TYPE not equal to 0 as a result of an oversight; the drive should actually be simulated (= 0).
- $MA_CTRLOUT_MODULE_NR entered incorrectly, i.e. the logical drive numbers were exchanged and an invalid value is stored for this drive in $MN_DRIVE_LOGIC_ADDRESS (see 3.) or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the Profibus (i.e. the values are not in SDB1000) or different addresses were selected for the input and output slots of the drive in the Profibus configuration.

Program Continuation: Switch control OFF - ON.

26106 Encoder %2 of axis %1 not found

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

Definitions: The drive configured for the specified axis could not be found. For example, a Profibus slave was configured on the NC but is not contained in SDB1000.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Possible causes:
- $MA_ENC_TYPE not equal to 0 as a result of an oversight; the encoder should actually be simulated (= 0).
- $MA_ENC_MODULE_NR entered incorrectly, i.e. the logical drive numbers were exchanged and an invalid value is stored for this drive in $MN_DRIVE_LOGIC_ADDRESS (see next paragraph) or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the Profibus (i.e. the values are not in SDB1000) or different addresses were selected for the input and output slots of the drive in the Profibus configuration.

Program Continuation: Switch control OFF - ON.

26110 Independent drive stop/retract triggered

Definitions: Informational alarm: An "independent extended stop or retract" was triggered on the drive bus for at least one axis. The drive in question subsequently ignores NC travel commands. The bus must be rebooted (hardware reset).

Reactions:
- NC not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Reboot the drive, hardware reset.
Program Continuation: Switch control OFF - ON.

27000 Axis %1 is not safely referenced

Parameters: %1 = Axis number
Definitions: There are two reasons for this alarm:
- the machine position has not yet been acknowledged by the user,
- the machine position has not yet been verified through follow-up referencing.

Even if the axis is already referenced, there is no confirmation that referencing has supplied the correct result. For example, wrong results can occur if the axis was moved after the control was switched off, with the result that the standstill position saved prior to switching off is no longer correct. To make sure that this does not happen, the user must acknowledge the displayed actual position after the first referencing process.

When the user enable has first been set, follow-up referencing must be carried out each time the control is booted (with absolute encoders, this follow-up referencing is executed automatically). This procedure is carried out to verify the standstill position saved prior to switching off of the control.

Via the MD $MN_SAFE_ALARM_SUPPRESS_LEVEL (MD>=3), the alarm display can be set in such a way that the group alarm 27100 is displayed for all SI axes.

Reactions: - Alarm display.
Remedy: Move the axis to a known position, change to the "Referencing" mode and press the soft key "Enable". Check on the machine the positions displayed in the enable diagram. If these correspond to those expected at the known positions, confirm this via the toggle key. If the user enable has already been set, reference the axis again.

WARNING:

If the axis has not been safely referenced and the user has not confirmed, the following applies:
- The safe cams are not yet safe.
- The safe limit positions are not yet active.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27001 Axis %1 error in a monitoring channel, code %2, values: NCK %3, drive %4

Parameters: %1 = Axis number
%2 = Additional info cross-comparison index
%3 = NCK comparison value extension
%4 = Additional info comparison value drive

Definitions: The mutual comparison of the two monitoring channels has found a difference between input data or results of the monitoring operations. One of the monitors no longer functions reliably, i.e. safe operation is no longer possible.

The following error codes are possible on the NCK side:
- 0 Following alarm to drive alarm 300911.
- 1 result list 1: SBH, SG, SBR or SE result. For further information see drive MD 1391, 1392.
- 2 result list 2: SN, n_x result. For further information see drive MD 1393, 1394.
- 3 Actual value difference greater than setting in $MA_SAFE_POS_TOL.
- 4 Not assigned.
- 5 Function enables $MA_SAFE_FUNCTION_ENABLE.
- 6 Velocity limit $MA_SAFE_VELO_LIMIT[0].
- 7 Velocity limit $MA_SAFE_VELO_LIMIT[1].
- 8 Velocity limit $MA_SAFE_VELO_LIMIT[2].
- 9 Velocity limit $MA_SAFE_VELO_LIMIT[3].

© Siemens AG, 2002. All rights reserved

- 10 Tolerance for safe standstill $MA_SAFE_STANDSTILL_TOL.
- 11 Safe position limit $MA_SAFE_POS_LIMIT_PLUS[0].
- 12 Safe position limit $MA_SAFE_POS_LIMIT_MINUS[0].
- 13 Safe position limit $MA_SAFE_POS_LIMIT_PLUS[1].
- 14 Safe position limit $MA_SAFE_POS_LIMIT_MINUS[1].
- 15 Cam position $MA_SAFE_CAM_POS_PLUS[0] + $MA_SAFE_CAM_TOL.
- 16 Cam position $MA_SAFE_CAM_POS_PLUS[0].
- 17 Cam position $MA_SAFE_CAM_POS_MINUS[0] + $MA_SAFE_CAM_TOL.
- 18 Cam position $MA_SAFE_CAM_POS_MINUS[0].
- 19 Cam position $MA_SAFE_CAM_POS_PLUS[1] + $MA_SAFE_CAM_TOL.
- 20 Cam position $MA_SAFE_CAM_POS_PLUS[1].
- 21 Cam position $MA_SAFE_CAM_POS_MINUS[1] + $MA_SAFE_CAM_TOL.
- 22 Cam position $MA_SAFE_CAM_POS_MINUS[1].
- 23 Cam position $MA_SAFE_CAM_POS_PLUS[2] + $MA_SAFE_CAM_TOL.
- 24 Cam position $MA_SAFE_CAM_POS_PLUS[2].
- 25 Cam position $MA_SAFE_CAM_POS_MINUS[2] + $MA_SAFE_CAM_TOL.
- 26 Cam position $MA_SAFE_CAM_POS_MINUS[2].
- 27 Cam position $MA_SAFE_CAM_POS_PLUS[3] + $MA_SAFE_CAM_TOL.
- 28 Cam position $MA_SAFE_CAM_POS_PLUS[3].
- 29 Cam position $MA_SAFE_CAM_POS_MINUS[3] + $MA_SAFE_CAM_TOL.
- 30 Cam position $MA_SAFE_CAM_POS_MINUS[3].
- 31 Actual position tolerance $MA_SAFE_POS_TOL. $MA_SAFE_SLIP_VELO_TOL for active actual value synchronization (slippage)
- 32 Ref.position tolerance $MA_SAFE_REFP_POS_TOL.
- 33 Delay SG[x] -> SG[y] $MA_SAFE_VELO_SWITCH_DELAY.
- 34 Delay cross-comparison $MA_SAFE_MODE_SWITCH_TIME.
- 35 Delay pulse disable Stop B $MA_SAFE_PULSE_DISABLE_DELAY.
- 36 Delay pulse disable test stop $MA_SAFE_PULSE_DIS_CHECK_TIME
- 37 Delay Stop C -> SBH $MA_SAFE_STOP_SWITCH_TIME_C.
- 38 Delay Stop D -> SBH $MA_SAFE_STOP_SWITCH_TIME_D.
- 39 Delay Stop E -> SBH $MA_SAFE_STOP_SWITCH_TIME_E.
- 40 Stop reaction on SG exceeded $MA_SAFE_VELO_STOP_MODE.
- 41 Stop reaction on SE exceeded $MA_SAFE_POS_STOP_MODE.
- 42 Standstill speed $MA_SAFE_STANDSTILL_VELO_TOL.
- 43 Memory test, stop reaction.
- 44 Actual position + SG[0] $MA_SAFE_VELO_LIMIT[0].
- 45 Actual position - SG[0] $MA_SAFE_VELO_LIMIT[0].
- 46 Actual position + SG[1] $MA_SAFE_VELO_LIMIT[1].
- 47 Actual position - SG[1] $MA_SAFE_VELO_LIMIT[1].
- 48 Actual position + SG[2] $MA_SAFE_VELO_LIMIT[2].
- 49 Actual position - SG[2] $MA_SAFE_VELO_LIMIT[2].
- 50 Actual position + SG[3] $MA_SAFE_VELO_LIMIT[3].
- 51 Actual position - SG[3] $MA_SAFE_VELO_LIMIT[3].
- 52 Standstill position + tolerance $MA_SAFE_STANDSTILL_TOL.
- 53 Standstill position - tolerance $MA_SAFE_STANDSTILL_TOL.
- 54 Actual position + n_x + tolerance $MA_SAFE_VELO_X + $MA_SAFE_POS_TOL.
- 55 Actual position + n_x $MA_SAFE_VELO_X.
- 56 Actual position - n_x $MA_SAFE_VELO_X.
- 57 Actual position - n_x - tolerance $MA_SAFE_VELO_X - $MA_SAFE_POS_TOL
- 58 Active external standstill request.

© Siemens AG, 2002. All rights reserved

- 59 SG override factor 1 $MA_SAFE_VELO_OVR_FACTOR[0].
- 60 SG override factor 2 $MA_SAFE_VELO_OVR_FACTOR[1].
- 61 SG override factor 3 $MA_SAFE_VELO_OVR_FACTOR[2].
- 62 SG override factor 4 $MA_SAFE_VELO_OVR_FACTOR[3].
- 63 SG override factor 5 $MA_SAFE_VELO_OVR_FACTOR[4].
- 64 SG override factor 6 $MA_SAFE_VELO_OVR_FACTOR[5].
- 65 SG override factor 7 $MA_SAFE_VELO_OVR_FACTOR[6].
- 66 SG override factor 8 $MA_SAFE_VELO_OVR_FACTOR[7].
- 67 SG override factor 9 $MA_SAFE_VELO_OVR_FACTOR[8].
- 68 SG override factor 10 $MA_SAFE_VELO_OVR_FACTOR[9].
- 69 SG override factor 11 $MA_SAFE_VELO_OVR_FACTOR[10].
- 70 SG override factor 12 $MA_SAFE_VELO_OVR_FACTOR[11].
- 71 SG override factor 13 $MA_SAFE_VELO_OVR_FACTOR[12].
- 72 SG override factor 14 $MA_SAFE_VELO_OVR_FACTOR[13].
- 73 SG override factor 15 $MA_SAFE_VELO_OVR_FACTOR[14].
- 74 SG override factor 16 $MA_SAFE_VELO_OVR_FACTOR[15].
- 75 Velocity limit n_x $MA_SAFE_VELO_X.
- 76 Stop reaction SG1 $MA_SAFE_VELO_STOP_REACTION[0].
- 77 Stop reaction SG2 $MA_SAFE_VELO_STOP_REACTION[1].
- 78 Stop reaction SG3 $MA_SAFE_VELO_STOP_REACTION[2].
- 79 Stop reaction SG4 $MA_SAFE_VELO_STOP_REACTION[3].
- 80 Modulo value for safe cam $MA_SAFE_MODULO_RANGE.
- 81 Velocity tolerance for safe deceleration ramp $MA_SAFE_STOP_VELO_TOL.
- 82 SG override factor SGE 0...15 = active SGE position. -1 = SG override inactive (neither SG2 nor SG4 active, or function not activated in $MA_SAFE_FUNCTION_ENABLE).
- 83 Acceptance test time different $MA_SAFE_ACCEPTANCE_TEST_TIMEOUT.
- 84 Delay time Stop F -> Stop B $MA_SAFE_STOP_SWITCH_TIME_F.
- 85 Delay time pulse disable bus fail $MN_SAFE_PULSE_DIS_TIME_BUSFAIL.
- 86 Not assigned.
- 87 Not assigned.
- 88 Not assigned.
- 89 Encoder limit frequency $MA_SAFE_ENC_FREQ_LIMIT (Performance_2 only).
- 1000 Control timer expired: If one channel informs another of an SGE change, this control timer is used to check whether the update timer in the other channel has expired.
- 1001 (only assigned on drive, see alarm 300911)
- 1002 User confirmation inconsistent: Data for user confirmation different in both monitoring channels after 2 seconds.
- 1003 Reference tolerance exceeded.
- 1004 Plausibility error in user confirmation.
- 1005 Pulses already disabled on test stop selection.
- 1006 (only assigned on drive, see alarm 300911).
- 1007 (only assigned on drive, see alarm 300911).
- 1008 (only assigned on drive, see alarm 300911).
- 1009 Pulses not disabled after $MA_SAFE_PULSE_DIS_CHECK_TIME test stop time.
- 1010 Pulses not disabled with a test of the external pulse suppression after $MA_SAFE_PULSE_DIS_CHECK_TIME test stop time.
- 1011 NCK/drive acceptance test state different.
- 1020 Communication disrupted between NCK monitoring channel and drive monitoring channel.

© Siemens AG, 2002. All rights reserved

Reactions: - NC Start disable in this channel.
- Alarm display.

Remedy: STOP F is enabled, that means a message is generated and NC start interlock is set (if only data cross-comparison is activated), or, with active monitoring, the immediate switchover to STOP B. Display on the NC/PLC interface occurs.

Find the difference between the monitoring channels. The error code indicates the cause. A possible cause is that the safety-related machine data are no longer identical or that the SGEs do not have the same level (recalibrate). If no error of this type is apparent, an error may have occurred in the CPU, e.g. a "flipped" memory cell. This error can be temporary (in this case it can be eliminated by a Power On) or permanent (if it occurs again after Power On replace the hardware).

If safe monitoring was active, STOP B was also enabled automatically. It is necessary to switch the control off/on (POWER ON).

Error codes for STOP F for 840D/611D:

0: No error. Definition: There is no error in this channel. An error may have occurred in the other channel. Cause, remedy: Search the cause in the other channel and interprete the error code.

1: Results list. Unequal control of the functions via the SGEs. Evaluate precise error coding in the 611D-MD 1391 and 1392.

2: Results list. Check cam tolerance, evaluate precise error coding in the 611D-MD 1393 and 1394.

3: Actual position. Incorrect encoder evaluation (check MDs). Differently stored standstill position.

4: No cross-comparison.

5: Function enables. Enter equal MDs.

6: Limit value for SG1. Enter equal MDs.

7: Limit value for SG2. Enter equal MDs.

8: Limit value for SG3. Enter equal MDs.

9: Limit value for SG4. Enter equal MDs.

10: Standstill tolerance. Enter equal MDs.

11: Upper limit value SE1. Enter equal MDs.

12: Lower limit value SE1. Enter equal MDs.

13: Upper limit value SE2. Enter equal MDs.

14: Lower limit value SE2. Enter equal MDs.

15: Safe cam 1+ (+tolerance). Enter equal MDs.

16: Safe cam 1+. Enter equal MDs.

17: Safe cam 1- (+tolerance). Enter equal MDs.

18: Safe cam 1-. Enter equal MDs.

19: Safe cam 2+ (+tolerance). Enter equal MDs.

20: Safe cam 2+. Enter equal MDs.

21: Safe cam 2- (+tolerance). Enter equal MDs.

22: Safe cam 2-. Enter equal MDs.

23: Safe cam 3+ (+tolerance). Enter equal MDs.

24: Safe cam 3+. Enter equal MDs.

25: Safe cam 3- (+tolerance). Enter equal MDs.

26: Safe cam 3-. Enter equal MDs.

27: Safe cam 4+ (+tolerance). Enter equal MDs.

28: Safe cam 4+. Enter equal MDs.

29: Safe cam 4- (+tolerance). Enter equal MDs.

30: Safe cam 4-. Enter equal MDs.

31: Position tolerance. Enter equal MDs.

32: Reference position tolerance. Enter equal MDs.

© Siemens AG, 2002. All rights reserved

33: Time velocity changeover. Enter equal MDs.
34: Tolerance time SGE changeover. Enter equal MDs.
35: Delay time pulse deletion. Enter equal MDs.
36: Time for check of pulse deletion. Enter equal MDs.
37: Transition time STOP C to SBH. Enter equal MDs.
38: Transition time STOP D to SBH. Enter equal MDs.
39: Transition time STOP E to SBH. Enter equal MDs.
40: Stop reaction to SG. Enter equal MDs.
41: Stop reaction to SE. Enter equal MDs.
42: Creep speed pulse deletion. Enter equal MDs.
43: Storage test stop reaction.
44: Actual position value + limit value SG1.
45: Actual position value - limit value SG1.
46: Actual position value + limit value SG2.
47: Actual position value - limit value SG2.
48: Actual position value + limit value SG3.
49: Actual position value - limit value SG3.
50: Actual position value + limit value SG4.
51: Actual position value - limit value SG4.
52: Standstill position + tolerance.
53: Standstill position - tolerance.
54: Actual position value "+ nx" + tolerance.
55: Actual position value "+ nx".
56: Actual position value "- nx".
57: Actual position value "- nx" + tolerance.
58: Current shutdown request.
59: SG override factor 1. Enter equal MDs.
60: SG override factor 2. Enter equal MDs.
61: SG override factor 3. Enter equal MDs.
62: SG override factor 4. Enter equal MDs.
63: SG override factor 5. Enter equal MDs.
64: SG override factor 6. Enter equal MDs.
65: SG override factor 7. Enter equal MDs.
66: SG override factor 8. Enter equal MDs.
67: SG override factor 9. Enter equal MDs.
68: SG override factor 10. Enter equal MDs.
69: SG override factor 11. Enter equal MDs.
70: SG override factor 12. Enter equal MDs.
71: SG override factor 13. Enter equal MDs.
72: SG override factor 14. Enter equal MDs.
73: SG override factor 15. Enter equal MDs.
74: SG override factor 16. Enter equal MDs.
75: Velocity limit "nx". Enter equal MDs.
76: Stop reaction with SG1. Enter equal MDs.
77: Stop reaction with SG2. Enter equal MDs.
78: Stop reaction with SG3. Enter equal MDs.
79: Stop reaction with SG4. Enter equal MDs.
80: Modulo value for safe cams. Enter equal MDs.
81: Velocity tolerance for safe braking ramp. Enter equal MDs.
1000: Control timer expired. E.g., contact problems (loose contact).

© Siemens AG, 2002. All rights reserved

1001: Incorrect control timer initialization. Cause, remedy: -
1002: User confirmation timer expired.
1003: Reference tolerance affected.
1004: Plausibility of user confirmation is affected.
1005: Pulses already deleted during test stop selection. Test stop selection with missing pulse enable, error during wiring of the SGE "pulses have been deleted".
1006: Error during forced SGA dynamization.
1007: Communication failure between PLC and drive.
1008: Erroneous data transfer between PLC and drive.
1009: Trigger a subsequent stop after test stop.
1010: Pulses not deleted.
1020: Cyclic communication between NCK and drive no longer functions.

Program Continuation: Clear alarm with the RESET key. Restart part program

27002 Axis %1 test stop is running

Parameters: %1 = Axis number

Definitions: Proper functioning of the switch-off path is just being tested by setting of the SGE "Test stop selection".

Reactions: - Alarm display.

Remedy: The message serves only for user information.
The alarm automatically disappears after expiration of the delay time that is defined in MD $MA_SAFE_PULSE_DIS_CHECK_TIME, and the removal of the SGE "Test stop selection" when the controller detects pulse deletion, i.e., the test is successfully concluded. A successful test can be recognized by the alarm 27001 with the error code 1005 or the alarm 27024.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27003 Checksum error found: %1 %2

Parameters: %1 = Note on code section or table
%2 = Table number

Definitions: Checksum error in safety-relevant code or safety-relevant data. The safe monitoring functions (Safety Integrated) in the NCK could be affected.

Reactions: - Alarm display.

Remedy: Continue to work very carefully. Reload code and data as soon as possible (Power On). If this error occurs again, contact your service personnel.

Program Continuation: Switch control OFF - ON.

27004 Axis %1, difference safe input %2, NCK %3, drive %4

Parameters: %1 = Axis number
%2 = Monitoring input
%3 = Interface identifier NCK input
%4 = Interface identifier drive input

Definitions: A difference has been found on the specified safe input. The state of the specified input signal differed in the two monitoring channels NCK and 611D during the duration set in $MA_SAFE_MODE_SWITCH_TIME.
Monitoring in question:
SS/SV = Difference in SGE "Deselection of safe operating stop/Safe velocity"
SS = Difference in SGE "Safe operating stop"
SV = Difference in SGEs "Selection safe velocity"
SP = Difference in SGE "Selection safe end position"

© Siemens AG, 2002. All rights reserved

SVOVR = Difference in SGEs "Selection SG correction"
Interface identifier NCK input:
DMP<drv><mod><bit>=<value>
<drv> = Drive number of terminal block (1...31)
<mod> = Submodule number (1...8)
<bit> = Terminal number (1...16)
<value> = Value of NCK SGE (0,1)
SPL For when the SGE is parameterized at the SPL interface.
<io> = Parameterizable system variable range (01=$A_INSID, 02=$A_INSED)
<dword> = System variable double word (1,2)
<bit> = Bit number in system variable double word (1...32)
<value> = Value of NCK SGE (0,1)
Onboard input For when the SGE is parameterized at an onboard input.
<bit> = Input number = 01 ...04
<value> = Value of NCK SGE = 0,1
Interface identifier drive input:
DBX<byte><bit>=<value >
<byte> = Byte number in axial DB (22, 23, 32, 33)
<bit> = Bit number in byte (0...7)
<value> = Value of drive SGE (0,1)
This alarm can be hidden by setting MD $MN_SAFE_DIAGNOSIS_MASK, bit 0 = 0.

Reactions: - Alarm display.

Remedy: Check settings for safe input signals (NCK I/Os, PLC DB parameters).

Program Continuation: Clear alarm with the RESET key. Restart part program

27005 Axis %1 error in data cross check: static actual value difference

Parameters: %1 = Axis number

Definitions: Via the data cross check between NCK and 611D monitoring channel, a difference in actual values was detected, which is greater than the maximum tolerance defined in MD $MA_SAFE_POS_TOL. This can be checked by means of the safe position values for the two monitoring channels displayed in the service menu.
The alarm is displayed only, if monitoring with absolute reference (SE/SN) has been enabled for the specified axis and if the user enable has been set. The alarm is cleared, as soon as the user enable is deleted or the actual value difference between the two monitoring channels falls again below the maximum permissible difference.

Reactions: - Alarm display.

Remedy: If the alarm is present statically, the user enable must be deleted. When the control is then rebooted, the machine can be brought to the safe state again and operation resumed by a new referencing process and setting of the user enable. Prior to setting the user enable, the actual position of the axis displayed in the "User enable" screen must be compared with the current machine position. This is obligatory to ensure the ensure proper functioning of the safe limit positions (SE) and safe cams (SN).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27006 Axis %1 Test ext. pulse deletion running

Parameters: %1 = Axis number

Definitions: The correct functioning of the external pulse disable is being checked now by setting the "Test stop of external shutdown" SGE.

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Alarm disappears automatically when the test has been exited by deleting the "Test stop of external shutdown" SGE.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27007 Axis %1 acceptance test mode is active

Parameters: %1 = Axis number

Definitions: Via the operator panel, an SI acceptance test has been started with the acceptance test wizard. The acceptance test mode is activated via the NCK and drive for the time of this acceptance test. In the acceptance test mode, SI PowerOn alarms can be acknowledged with the Reset key.

Reactions: - Alarm display.

Remedy: Deselect the acceptance test with the acceptance test wizard or wait until completed (acceptance test time can be parameterized via MD $MA_SAFE_ACCEPTANCE_TEST_TIMEOUT).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27008 Axis %1 SW limit switch deactivated

Parameters: %1 = Axis number

Definitions: Via the operator panel, an SI acceptance test safe end position has been started with the acceptance test wizard. For these acceptance tests, the single-channel software limit switches are deactivated for the axis/spindle.

Reactions: - Alarm display.

Remedy: Deselect the acceptance test with the acceptance test wizard or wait until completed.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27010 Axis %1 tolerance for safe standstill exceeded

Parameters: %1 = Axis number

Definitions: The axis has moved too far away from the setpoint position. It is further away than allowed in MD SAFE_STANDSTILL_TOL.

STOP of NC program processing. Stop of the axis with speed setpoint = 0 (STOP B). As soon as the actual speed value is less than it is defined in MD $MA_SAFE_STANDSTILL_VELO_TOL, but no later than after the expiration of the time in MD $MA_SAFE_PULSE_DISABLE_DELAY.

The alarm can be reprogrammed in the MD $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Channel not ready.

Remedy: Check the tolerance of zero speed control: does the value match the precision and control dynamics of the axis? If not, increase tolerance. If yes, check the machine for damage and remove it.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

27011 Axis %1 safe velocity exceeded

Parameters: %1 = Axis number

Definitions: The axis has moved too quickly and faster than allowed in MD $MA_SAFE_VELO_LIMIT. With active SBH/SG and a 1-encoder system, the velocity which corresponds to an encoder limit frequency of 200 kHz has been exceeded.

Stop of the axis with STOP C, D or E, according to the configuration in MD $MA_SAFE_VELO_STOP_MODE. Blocking of NC START. Display on the interface.

Reactions:
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. If no obvious operator error occurred: Check input value of the MD, Check SGEs: was the correct safe velocity selected? If the MDs and SGEs are o.k., check the machine for any damage and rectify.

Program Continuation: Clear alarm with the RESET key. Restart part program

27012 Axis %1 safe end position exceeded

Parameters: %1 = Axis number

Definitions: The axis has exceeded the limit position entered in MD $MA_SAFE_POS_LIMT_PLUS or MD $MA_SAFE_POS_LIMIT_MINUS.

Stop of the axis with STOP C, D or E, according to the configuration in MD: $MA_SAFE_POS_STOP_MODE. Blocking of NC START. Display on the NC/PLC interface.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. If no obvious operator error occurred: Check the input value of the machine data and check the SGEs: was the correct one of 2 end positions selected? If the MDs and SGEs are o.k., check the machine for any damage and rectify.

Remove the user confirmation for this axis. Actuate the RESET key, the program will be aborted and the alarm will be deleted. Move the axis in the JOG mode to the valid traversing range. After eliminating the NC program error and checking the position of this axis, the user confirmation can be given again and the program can be restarted.

Program Continuation: Clear alarm with the RESET key. Restart part program

27013 Axis %1 safe braking ramp exceeded

Parameters: %1 = Axis number

Definitions: After the initiation of STOP B or C, the velocity exceeded the tolerance value entered in MD $MA_SAFE_STOP_VELO_TOL.

Pulse blocking (via SGA).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the MD. Check the braking behavior of the affected drive.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

27020 Axis %1 stop E triggered

Parameters: %1 = Axis number

Definitions: This alarm comes with the alarms 27011 "Safe velocity exceeded" or 27012 "Safe end position exceeded" (when configured as such in MD $MA_SAFE_VELO_STOP_MODE or MD: $MA_SAFE_POS_STOP_MODE). It indicates that a LIFTFAST-ASUP has been triggered and the internal activation of the "Safe operating stop".

Stop of the axis with STOP E.

Reactions:
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Eliminate causes of the alarms "Safe velocity exceeded" or "Safe end position exceeded" (see description of these alarms).

Program Continuation: Clear alarm with the RESET key. Restart part program

27021 Axis %1 stop D triggered

Parameters: %1 = Axis number

Definitions: This alarm comes with the alarms 27011 "Safe velocity exceeded" or 27012 "Safe end position exceeded" (when configured as such in $MA_SAFE_VELO_STOP_MODE or $MA_SAFE_POS_STOP_MODE). It indicates that a "Deceleration on the path" has been triggered and the internal activation of the "Safe operating stop".

Stop of the axis with STOP D.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Eliminate causes of alarm "Safe velocity exceeded" or "Safe end position exceeded" (see description of these alarms).

Program Continuation: Clear alarm with the RESET key. Restart part program

27022 Axis %1 stop C triggered

Parameters: %1 = Axis number

Definitions: This alarm comes with the alarms 27011 "Safe velocity exceeded" or 27012 "Safe end position exceeded" (when configured as such in $MA_SAFE_VELO_STOP_MODE or $MA_"SAFE_POS_STOP_MODE). It indicates that a "Deceleration on the current limit" has been triggered and the internal activation of the "Safe operating stop".

Stop of the axis with STOP C.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Eliminate causes of alarm "Safe velocity exceeded" or "Safe end position exceeded" (see description of these alarms).

Program Continuation: Clear alarm with the RESET key. Restart part program

27023 Axis %1 stop B triggered

Parameters: %1 = Axis number

Definitions: This alarm comes with the alarm 27010 "Tolerance for safe standstill exceeded" or 27001 "STOP F triggered". It indicates that a "Deceleration on the current limit" has been triggered and the internal activation of the timer for changeover to STOP A (see MD $MA_SAFE_PULSE_DISABLE_DELAY).

© Siemens AG, 2002. All rights reserved

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Stop of the axis with STOP B. Start of the switchover timer after STOP A.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Eliminate causes of alarm "Safe velocity exceeded" or "Safe end position exceeded" (see description of these alarms).

Program Continuation: Switch control OFF - ON.

27024 Axis %1 stop A triggered

Parameters: %1 = Axis number

Definitions: This alarm comes with the alarm 27011 "Safe velocity exceeded" (when configured as such in $MA_SAFE_VELO_STOP_MODE) or follows from STOP B or an unsuccessful test stop. It indicates that a "Cancel pulse" has been triggered.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Stop of the axis with STOP A. Pulse delete (via SGA).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Eliminate causes of "Safe velocity exceeded" or for the previous STOP B (see description of these alarms).

Program Continuation: Switch control OFF - ON.

27030 Axis %1 function not supported on this 611D module

Parameters: %1 = Axis number

Definitions: SINUMERIK Safety Integrated can be used only with the 611D Performance control modules with 2 measuring circuits per drive and cutoff relay. An attempt has been made to activate a safety function although no such module is plugged in.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Replace module or switch off safety functions in MD $MA_SAFE_FUNCTION_ENABLE.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

27031 Axis %1 limit value for safe velocity %2 at gear ratio %3 too large (max. %4)

Parameters: %1 = Axis number
%2 = Limit value index
%3 = Number of the transmission ratio
%4 = Maximum velocity

Definitions: All limit values in MD $MA_SAFE_VELO_LIMIT have to be so set, that the limit frequency of the limiting frequency of the amplitude monitoring in the measuring circuit hardware (200 kHz , from SW 4.2 300 kHz up) is not exceeded. The limit value which did not fulfil this condition is indicated as second parameter (1 for SG1, 2 for SG2, etc.). The third parameter indicates the gear stage, e.g. 1 for gear stage 1, 2 for gear stage 2, etc. The fourth parameter indicates the maximum velocity which can be entered to just maintain the limit frequency in safe operation.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).
Pulse blocking (via SGA).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Reduce the limit value in MD $MA_SAFE_VELO_LIMIT[x], x = (2nd alarm parameter) - 1, or correct the setting of the gear factors.

Program Continuation: Switch control OFF - ON.

27032 Axis %1 checksum error of safe monitoring. Confirmation and re-test required!

Parameters: %1 = Axis number

Definitions: The machine data "MA_SAFE_..." are protected by a checksum after the control has been accepted. The alarm indicates that the current checksum does not coincide any longer with the stored checksum, that means that either an MD value has been changed without authorization or a MD is defective.
Pulse blocking (via SGA).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check MDs. Allow the checksum to be recalculated. Re-accept safety functions.

Program Continuation: Switch control OFF - ON.

27033 Axis %1 parameterization of MD %2[%3] invalid

Parameters: %1 = Axis number
%2 = MD identifier
%3 = Machine data index

Definitions: The parameterization of %2 is incorrect. An additional indication is the field index of the machine data. If the machine data is a single machine data a zero is specified as array index. This alarm occurs in the following contexts:

© Siemens AG, 2002. All rights reserved

- 1. The conversion of the specified MD into the internal calculation format leads to overflow.
- 2. The values entered in MD $MA_SAFE_POS_LIMIT_PLUS and $MA_SAFE_POS_LIMIT_MINUS have been interchanged. The upper limit is less than or equal to the lower limit.
- 3. For an axis with safety functions, the setpoint/actual value assignment in MD $MA_SAFE_ENC_SEGMENT_NR, MD $MA_CTRLOUT_SEGMENT_NR was not made on the drive bus. No module number was stated for a setpoint/actual value assignment in MD $MA_CTRLOUT_MODULE_NR, MD $MA_SAFE_ENC_MODULE_NR.
- 4. The number of drives has changed. On reading back the standstill position and the associated drive number, a difference has been found to the momentary drive configuration.
- 5. A safety function has been enabled in MD $MA_SAFE_FUNCTION_ENABLE without the safety functions SBH/SG having been enabled.
- 6. Error on parameterizing the input/output settings for the SGEs/SGAs.
- 7. A zero has been entered in MD $MA_SAFE_ENC_GRID_POINT_DIST.
- 8. A zero has been entered in MD $MA_SAFE_ENC_RESOL .
- 9. In MD $MA_IS_ROT_AX and MD $MA_SAFE_IS_ROT_AX , differing settings were made.
- 10. A measuring circuit that does not exist was parameterized in MD $MA_SAFE_ENC_INPUT_NR.
- 11. In MD $MA_SAFE_ENC_MODULE_NR, the number of a drive has been entered which either does not exist or has been detected as inactive. With an inactive drive, MD $MA_SAFE_ENC_TYPE was not reset to 0.
- 12. In MD $MA_SAFE_ENC_TYPE, an encoder type was parameterized which does not match the physically present type.
- 13. In MD $MA_SAFE_ENC_TYPE, an incorrect encoder type was entered for an active drive ($MA_SAFE_ENC_TYPE = 0, 2, 3 or 5).
- 14. When setting the parameters for the motor encoder in MD $MA_SAFE_ENC_INPUT_NR, the measuring circuit for the 2nd measuring system is also used to ensure double-redundancy. The 2nd measuring circuit of this drive module has also been parameterized in the data of another axis, therefore there is a dual assignment. The 2nd measuring circuit connection cannot be used for the actual value acquisition in this parameterization.
- 15. In MD $MA_SAFE_POS_TOL a value greater than 10 mm was entered for a linear axis.
- 16. In MD $MA_SAFE_REFP_POS_TOL, a value greater than 1mm was entered for a linear axis.
- 17. The limit values for the "n<n_x" monitoring, calculated from MD $MA_SAFE_VELO_X and MD $MA_SAFE_POS_TOL, are of equal size.
- 18. One of the activated cam positions is outside the actual value modulo range.
- 19. The parameterized cam modulo range MD $MA_SAFE_MODULO_RANGE is not a multiple integer of 360 degrees.
- 20. The parameterized cam modulo range MD $MA_SAFE_MODULO_RANGE and the modulo range in MD $MA_MODULO_RANGE cannot be divided as integers into one another.
- 21. The "actual value synchronization 2-encoder system" function (slippage) is selected for a single-encoder system, or a function with an absolute reference (SE/SN) is active at the same time.
- 22. Alarms 27000/300950 should be suppressed for parking (MD $MA_SAFE_PARK_ALARM_SUPPRESS!=0). The SGA "Axis safely referenced" must be configured in MD $MA_SAFE_REFP_STATUS_OUTPUT.

© Siemens AG, 2002. All rights reserved

- 23. An axial SGE/SGA was configured at the SPL interface (segment number = 4) and the function enable for the external stops (MD $MA_SAFE_FUNCTION_ENABLE, bit 6) is missing.
- 24. An axial SGE/SGA was parameterized at the SPL interface (segment number = 4) and the SGE "Deselect ext. Stop A" (assignment via MD $MA_SAFE_EXT_STOP_INPUT[0]) was parameterized inverted (bit 31 = 1) or the SGE "Deselect ext. Stop A" was not parameterized at the SPL interface $A_OUTSI.
- 25. The function "Save actual value with incremental encoder" is enabled in MD $MA_ENC_REFP_STATE for the parameterizable incremental encoder, and a monitoring function with absolute reference (SE/SN) is enabled in MD $MA_SAFE_FUNCTION_ENABLE. It is not permissible to combine these functions.
- 26. A value greater than 1000 mm/min was entered for a linear axis in MD $MA_SAFE_STANDSTILL_VELO_TOL.
- 27. A value greater than 20000 mm/min was entered for a linear axis in MD $MA_SAFE_STOP_VELO_TOL.
- 28. A value greater than 1000 mm/min was entered for a linear axis in MD $MA_SAFE_VELO_X.
- 29. A value greater than 1000 mm/min was entered for a linear axis in $MA_SAFE_SLIP_VELO_TOL.
- 30. A value greater than the maximum settable encoder limit frequency for the safe operation of a single-encoder system was set in MD $MA_SAFE_ENC_FREQ_LIMIT.
- 31. A value greater than 300 kHz for a Performance-1 or Standard-2 control module was set in MD $MA_SAFE_ENC_FREQ_LIMIT.
- 32. MD $MA_SAFE_EXT_PULSE_ENAB_OUTPUT was not or not correctly parameterized. A parameterization of this MD is required if in MD $MA_SAFE_PULSE_ENABLE_OUTPUT, bit 30 is set to 1, i.e. internal pulse suppression is being used.
- 33. The MD $MN_SAFE_SPL_STOP_MODE has been parameterized to the value of 4 (Stop E) without having enabled the external Stop E in all the axes with SI function enables (MD $MA_SAFE_FUNCTION_ENABLE not equal to 0).
- 34. Testing the mechanical system of the brakes was enabled in MD $MA_FIXED_STOP_MODE (bit 1 = 1), without previously enabling the safe operation function for this axis in MD $MA_SAFE_FUNCTION_ENABLE. Testing the mechanical system of the brakes is only admitted with safety functions in this axis.
- 35. Illegal values have been parameterized in MD $MA_SAFE_VELO_STOP_MODE or MD $MA_SAFE_VELO_STOP_REACTION.

As of SW 6.3, this alarm will also occur when:

- $MA_SAFE_EXT_PULSE_ENABLE_OUTPUT was not or not correctly parameterized. A parameterization of this MD is required if in MD $MA_SAFE_PULSE_ENABLE_OUTPUT, bit 30 is set to 1, i.e. internal pulse suppression is being used.
- A value greater than the maximum settable encoder limit frequency for the safe operation of a single-encoder system was set in $MA_SAFE_ENC_FREQ_LIMIT.
- A value greater than 300 kHz for a Performance-1 or Standard-2 control module was set in $MA_SAFE_ENC_FREQ_LIMIT.
- The MD $MN_SAFE_SPL_STOP_MODE has been parameterized to the value of 4 (Stop E) without having enabled the external Stop E in all the axes with SI function enables ($MA_SAFE_FUNCTION_ENABLE not equal to 0).

Reactions:

- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Check and change the stated MD. Allow the checksum to be recalculated. Re-accept safety functions. Parameterize MD $MN_SAFE_SPL_STOP_MODE to another stop mode or enable the external Stop E in the specified axes (set bits 4 and 6 in $MA_SAFE_FUNCTION_ENABLE).

The alarm is triggered during startup. No program can be started.

Program Continuation: Switch control OFF - ON.

27034 Parameterization of MD %1 invalid

Parameters: %1 = MD identifier

Definitions: The parameterization of %1 is incorrect. This alarm occurs in the following context: an incorrect value was entered for MD $MN_SAFE_ALARM_SUPPRESS_LEVEL.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the specified machine data.

Program Continuation: Switch control OFF - ON.

27090 Error in data cross check NCK-PLC, %1[%2], NCK: %3; %4<ALSI>

Parameters: %1 = Name of system variable in which the error was detected

%2 = System variable array index extension

%3 = NCK comparison value extension

%4 = Cross-check array index extension

Definitions: Error in the external wiring of the SPL terminals results in the previously issued alarm with information about the faulty system variables $A_INSE[1...64]. Via the MD $MN_SAFE_IN_HW_ASSIGN[0...7], the machine operator has to search for the module over which the system variable is supplied.

To simplify the diagnosis, the specification of the affected system variables in alarm parameter %1 is extended.

In addition to the names of the affected system variables ($A_INSE), the hardware assignment parameterized in MD $MN_SAFE_IN_HW_ASSIGN[0...7] is displayed, so that the affected HW connection can be determined from the information in the alarm line.

This extension is only made when there is a data cross check error at the $A_INSE system variables.

Example: Error in data cross check NCK PLC,

DMP 04.03 bit 01=$A_INSE[2], NCK: 1; 2

The information in the example (04.03) corresponds to the entries made in the MD $MN_SAFE_IN_HW_ASSIGN[0...7] about the system variables. They state:

DMP 04.03 bit 01The drive number of the affected terminal block (value range = 01...21)

DMP 04.03 bit 01Module number of the input module (value range = 01...08)

The stated numbers are hexadecimally displayed as in the MD $MN_SAFE_IN_HW_ASSIGN[0...7].

The bit-number specification begins, as does the numbering of the inputs on the DMP modules, with the value 0:

DMP 04.03 bit 012nd terminal (value range = 00... 15)

When assigning the SPL inputs to the NC onboard inputs, the extended alarm text looks like this:

Error in data cross check NCK PLC, NC onboard input 01=$A:INSE[1], NCK: 1; 2

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Find the difference between the monitoring channels. Possible causes:
- Incorrect wiring
- Incorrect SPL
- Incorrect assignment of axial SGEs to the internal interface $A_OUTSI
- Incorrect assignment of axial SGAs to the internal interface $A_INSI
- Incorrect assignment of SPL SGE's to the external interface $A_INSE
- Incorrect assignment of SPL SGA's to the external interface $A_OUTSE

Program Continuation: Clear alarm with the RESET key. Restart part program

27091 Error in data cross check NCK PLC, stop of %1

Parameters: %1 = Extension indicating the monitoring channel that triggered the stop

Definitions: The monitoring channel specified in %1 (NCK or PLC) has triggered a stop D or E (depending on the parameterization in MD $MN_SAFE_SPL_STOP_MODE). The alarm 27090 provides further information about the cause for the stop D/E.

Reactions: - Alarm display.

Remedy: Evaluate the alarm parameters of alarm 27090 and amend the SPL, or check the I/O modules/wiring or the internal SPL interfaces to the safety monitoring channels in the NCK and drive 611D.

Program Continuation: Clear alarm with the RESET key. Restart part program

27092 Communication broken off during NCK PLC data cross check, error detected by %1

Parameters: %1 = Extension indicating the monitoring channel that detected the error

Definitions: The delay time (10s) for communication monitoring was exceeded in the monitoring channel specified in %1 (NCK or PLC). The other monitoring channel did not send a new data packet within this time.

Reactions: - Alarm display.

Remedy: Do not start the SPL again. Check the system components (the PLC must have the correct version of FB15 and DB18).

Program Continuation: Switch control OFF - ON.

27093 Checksum error NCK-SPL, %1, %2, %3

Parameters: %1 = Extension indicating the type of error
%2 = Extension indicating the reference variable
%3 = Extension indicating the actual variable

Definitions: A checksum error has occurred in the NCK SPL. The file /_N_CST_DIR/_N_SAFE_SPF was subsequently modified. The safe programmable logic (SPL) in the NCK may be corrupted. Parameter %1 indicates the type of modification:
- %1 = FILE_LENGTH: the file length has changed.
- %1 = FILE_CONTENT: the file contents have changed.
- %2 specifies the reference variable (file length, checksum of file contents), %3 specifies the actual variable which is calculated cyclically.

Reactions: - Alarm display.

Remedy: Check the file and the time of the last modification to the file. Reload the original file and start the monitoring system again with a Power On.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

27094 Write access to system variable %1 only allowed from NCK-SPL

Parameters: %1 = Name of safety system variable concerned

Definitions: Write access to a safety system variable is only allowed from the part program / _N_CST_DIR/_N_SAFE_SPF. If this error occurs, an instruction from another part program was detected.

Reactions: - Alarm display.

Remedy: Check the part programs you are using for write accesses to safety system variables.

Program Continuation: Clear alarm with the RESET key. Restart part program

27095 %1 SPL protection not activated

Parameters: %1 = Name of the component on which the protection is not activated (NCK or PLC)

Definitions: The protection features are not activated for the SPL. The startup phase of the SPL is not yet complete. No stop reaction (Stop D) was initiated on an error in data cross-comparison between NCK and PLC.

Reactions: - Alarm display.

Remedy:
- Remedy for NCK: Activate the protection features with MD $MN_PREVENT_SYNACT_LOCK[0,1]. The number range of the synchronized action IDs used in the SPL must be entered in this MD.
- Remedy for PLC: Activate the protection features by setting the appropriate data bit in DB18.

Program Continuation: Clear alarm with the RESET key. Restart part program

27096 SPL start not allowed

Definitions: To start the SPL in protected state (MD $MN_PREVENT_SYNACT_LOCK[0,1] not equal 0) Safety Integrated functionality must first be activated for at least one axis (via MD $MA_SAFE_FUNCTION_ENABLE). Without this functionality it is only possible to operate SPL in start-up state.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Start up axial Safety Integrated functionality or remove the SPL protection via MD $MN_PREVENT_SYNACT_LOCK[0,1].

Program Continuation: Switch control OFF - ON.

27100 At least one axis is not safely referenced

Definitions: There are two reasons for this alarm:
- the machine position of at least one of the axes monitored with SI has not yet been acknowledged by the user, or
- the machine position of at least one of the axes monitored with SI has not yet been verified through follow-up referencing.

Even if the axis is already referenced, there is no confirmation that referencing has supplied the correct result. For example, wrong results can occur if the axis was moved after the control was switched off, with the result that the standstill position saved prior to switching off is no longer correct. To make sure that this does not happen, the user must acknowledge the displayed actual position after the first referencing process.

© Siemens AG, 2002. All rights reserved

When the user enable has first been set, follow-up referencing must be carried out each time the control is booted (with absolute encoders, this follow-up referencing is executed automatically). This procedure is carried out to verify the standstill position saved prior to switching off of the control.

Via the MD $MN_SAFE_ALARM_SUPPRESS_LEVEL (MD>=3), the alarm display can be set in such a way that an alarm is given for each axis individually which has not been safely referenced.

Reactions: - Alarm display.

Remedy: Move all SI axes to known positions and change to "Referencing" mode. Check the positions on the machine displayed in the user confirmation field and set "User confirmation" via the selection/toggle key. If the user confirmation for the axes has already been set, reference the axes again.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

27101 Axis %1 difference in function safe operational stop, NCK: %2 drive: %3

Parameters: %1 = Axis number

%2 = Monitoring status safe operating stop

%3 = Monitoring status safe operating stop

Definitions: During cross-comparison of result list 1 a difference was detected between the NCK and drive monitoring channels in the status of safe operating stop monitoring.

Safe operating stop: Bit 0,1 in result list 1

Monitoring status:

- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.

Remedy: Check whether the safe inputs have switched to the same status in both monitoring channels within the permissible time tolerance.

For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 1, NCK" and "Result list 1, drive" can be used.

Program Continuation: Clear alarm with the RESET key. Restart part program

27102 Axis %1 difference in function safe velocity %2, NCK: %3 drive: %4

Parameters: %1 = Axis number

%2 = SG level for which the difference was determined

%3 = Monitoring status safe velocity

%4 = Monitoring status safe velocity

Definitions: During cross-comparison of result list 1 a difference was detected between the NCK and drive monitoring channels in the status of safe velocity monitoring.

- Safe velocity 1: Bit 6, 17.78 cm result list 1
- Safe velocity 2: Bit 8, 22.86 cm result list 1
- Safe velocity 3: Bit 10, 27.94 cm result list 1
- Safe velocity 4: Bit 12, 33.02 cm result list 1

Monitoring status:

- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Check whether the safe inputs have switched to the same status in both monitoring channels within the permissible time tolerance.
For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 1, NCK" and "Result list 1, drive" can be used.
Program Continuation: Clear alarm with the RESET key. Restart part program

27103 Axis %1 difference in function safe limit position %2, NCK: %3 drive: %4

Parameters: %1 = Axis number
%2 = Number of safe limit position
%3 = Monitoring status safe limit position
%4 = Monitoring status safe limit position
Definitions: During cross-comparison of result list 1 a difference was detected between the NCK and drive monitoring channels in the status of safe limit position monitoring.
- Safe limit position 1: Bit 2, 7.62 cm result list 1
- Safe limit position 2: Bit 4, 12.70 cm result list 1

Monitoring status:
- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.
Remedy: Check whether the safe inputs have switched to the same status in both monitoring channels within the permissible time tolerance.
For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 1, NCK" and "Result list 1, drive" can be used.
Program Continuation: Clear alarm with the RESET key. Restart part program

27104 Axis %1 difference in function safe cam plus %2, NCK: %3 drive: %4

Parameters: %1 = Axis number
%2 = Cam number
%3 = Monitoring status safe cam plus
%4 = Monitoring status safe cam plus
Definitions: During cross-comparison of result list 2 a difference was detected between the NCK and drive monitoring channels in the status of safe cam plus monitoring.
- Safe cam 1+: Bit 0, 25.40 mm result list 2
- Safe cam 2+: Bit 4, 12.70 cm result list 2
- Safe cam 3+: Bit 8, 22.86 cm result list 2
- Safe cam 4+: Bit 12, 33.02 cm result list 2

Monitoring status:
- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.
Remedy: Check whether the safe inputs in both monitoring channels have switched to the same state within the permissible tolerance time.
For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 2, NCK" and "Result list 2, drive" can be used.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

27105 Axis %1 difference in function safe cam minus %2, NCK: %3 drive: %4

Parameters: %1 = Axis number
%2 = Cam number
%3 = Monitoring status safe cam minus
%4 = Monitoring status safe cam minus

Definitions: During cross-comparison of result list 2 a difference was detected between the NCK and drive monitoring channels in the status of safe cam minus monitoring.
- Safe cam 1-: Bit 2, 7.62 cm result list 2
- Safe cam 2-: Bit 6, 17.78 cm result list 2
- Safe cam 3-: Bit 10, 27.94 cm result list 2
- Safe cam 4-: Bit 14, 38.10 cm result list 2

Monitoring status:
- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.

Remedy: For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 2, NCK" and "Result list 2, drive" can be used.

Program Continuation: Clear alarm with the RESET key. Restart part program

27106 Axis %1 difference in function safe velocity nx, NCK: %2 drive: %3

Parameters: %1 = Axis number
%2 = Monitoring status safe velocity nx
%3 = Monitoring status safe velocity nx

Definitions: During cross-comparison of result list 2 a difference was detected between the NCK and drive monitoring channels in the status of safe velocity monitoring.
- Safe velocity nx+: Bit 16, 43.18 cm result list 2
- Safe velocity nx-: Bit 18, 48.26 cm result list 2

Monitoring status:
- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated
- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.

Remedy: For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 2, NCK" and "Result list 2, drive" can be used.

Program Continuation: Clear alarm with the RESET key. Restart part program

27107 Axis %1 difference in function cam modulo monitoring, NCK: %2 drive: %3

Parameters: %1 = Axis number
%2 = Monitoring status safe cam modulo range
%3 = Monitoring status safe cam modulo range

Definitions: During cross-comparison of result list 2 a difference was detected between the NCK and drive monitoring channels in the status of cam modulo monitoring.

Safe cam modulo range: Bit 20, 21 in result list 2

Monitoring status:
- OFF = Monitoring is inactive in this monitoring channel
- OK = Monitoring is active in this monitoring channel, limit values are not violated

© Siemens AG, 2002. All rights reserved

- L+ = Monitoring is active in this monitoring channel, upper limit exceeded
- L- = Monitoring is active in this monitoring channel, lower limit exceeded

Reactions: - Alarm display.

Remedy: Check whether the safe inputs have switched to the same status in both monitoring channels within the permissible time tolerance.
For further diagnostics, the drive machine data 1391, 1392 and the servo trace signals "Result list 2, NCK" and "Result list 2, drive" can be used.

Program Continuation: Clear alarm with the RESET key. Restart part program

27124 Stop A triggered at least in 1 axis

Definitions: This is only an informational alarm indicating that Stop A has been triggered in at least one axis and Power On is required for alarm acknowledgment.
This alarm occurs if the alarm priority function was activated in MD $MN_SAFE_ALARM_SUPPRESS_LEVEL.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Find the error cause by means of further alarm messages.

Program Continuation: Switch control OFF - ON.

27200 PROFIsafe: cycle time %1 [ms] too long

Parameters: %1 = Parameterized cycle time

Definitions: The PROFIsafe communication cycle time resulting from MD $MN_PROFISAFE_IPO_TIME_RATIO and MD $MN_IPO_CYCLE_TIME exceeds the permissible limit value (25 ms).

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Adapt cycle time via MD $MN_PROFISAFE_IPO_TIME_RATIO.

Program Continuation: Switch control OFF - ON.

27201 PROFIsafe: MD %1[%2]: bus segment %3 error

Parameters: %1 = MD name
%2 = MD field index
%3 = Parameterized bus segment

Definitions: An incorrect bus segment was entered in the specified machine data. The value must be 5.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct the MD.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

27202 PROFIsafe: MD %1[%2]: address %3 error

Parameters: %1 = MD name
%2 = MD field index
%3 = Parameterized PROFIsafe address

Definitions: An incorrect PROFIsafe address was entered in the specified machine data. The value must be greater than 0.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct the MD.

Program Continuation: Switch control OFF - ON.

27203 PROFIsafe: MD %1[%2]: SPL assignment error

Parameters: %1 = MD name
%2 = MD field index

Definitions: The parameterization of the specified machine data for the link between the SPL interface and a PROFIsafe module is incorrect because of the following reasons:
- Exchanged bit limits (upper bit value < lower bit value)
- Bit values greater than definition of SPL interface (bit value > 64)
- Number of bits too high for this PROFIsafe module (upper bit value - lower bit value + 1 > 8)
- No SPL assignment parameterized (both bit values equal to zero)
- Incorrect SPL assignment (bit value equals zero)

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct the MD.

Program Continuation: Switch control OFF - ON.

27204 PROFIsafe: double assignment MD %1[%2] - MD %3[%4]

Parameters: %1 = MD name 1
%2 = MD field index of MD name 1
%3 = MD name 2
%4 = MD field index of MD name 2

Definitions: A double assignment has illegally been parameterized in the specified machine data:
$A_INSE parameterized on DMP as well as PROFIsafe modules
- MD $MN_SAFE_IN_HW_ASSIGN
- MD $MN_PROFISAFE_IN_ASSIGN

$A_INSE parameterized on several PROFIsafe modules
- MD $MN_PROFISAFE_IN_ASSIGN

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Correct the MD.

Program Continuation: Switch control OFF - ON.

27220 PROFIsafe: Number of NCK F modules (%1) <> number of DP modules (%2)

Parameters: %1 = Number of parameterized NCK F modules
%2 = Number of parameterized S7 F modules

Definitions: The number of F modules parameterized via the NCK machine data $MN_PROFISAFE_IN/OUT_ADDRESS is:
- Greater than the number of PROFIBUS slaves in the S7 PROFIBUS configuration.
- smaller than the number of F modules in the S7 PROFIBUS configuration,
- greater than the number of F modules known in the S7 PROFIBUS configuration.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the F parameterization in the MD $MN_PROFISAFE_IN/OUT_ADDRESS.
Check the F configuration in the Step7 hardware configuration.

Program Continuation: Switch control OFF - ON.

27221 PROFIsafe: NCK F module MD %1[%2] unknown

Parameters: %1 = MD name
%2 = MD field index

Definitions: The F module parameterized in the specified machine data is unknown under this PROFIsafe address in the S7 PROFIBUS configuration.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the PROFIsafe addresses in the NCK MD and S7 PROFIBUS configuration.

Program Continuation: Switch control OFF - ON.

27222 PROFIsafe: S7 F module PROFIsafe address %1 unknown

Parameters: %1 = PROFIsafe address

Definitions: The F module with the specified PROFIsafe address has not been parameterized as an F module in the NCK MD.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Check the S7 PROFIBUS configuration. Enter the module in the NCK MD.
Program Continuation: Switch control OFF - ON.

27223 PROFIsafe: NCK F module MD %1[%2] is not a %3 module

Parameters: %1 = MD name
%2 = MD field index
%3 = Module type

Definitions: The F module parameterized in the specified NCK MD has not been entered as input/output module in the S7 PROFIBUS configuration.
- %3 = INPUT:NCK F parameterization expects INPUT module
- %3 = OUTPUT:NCK F parameterization expects OUTPUT module
- %3 = IN/OUT:NCK F parameterization expects INPUT or OUTPUT module

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the module in the S7 PROFIBUS configuration.
Program Continuation: Switch control OFF - ON.

27224 PROFIsafe: F module MD %1[%2] - MD %3[%4]: double assignment of PROFIsafe address

Parameters: %1 = MD name 1
%2 = MD field index 1
%3 = MD name 2
%4 = MD field index 2

Definitions: In the NCK MD or in the S7 F parameters, the same PROFIsafe address has been parameterized for the F modules parameterized in the specified machine data. Therefore, no clear communication link is possible between F master and F slave.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check and correct the S7 F parameterization and NCK MD.
Program Continuation: Switch control OFF - ON.

27225 PROFIsafe: slave %1, configuration error %2

Parameters: %1 = PROFIBUS slave address
%2 = Configuration error

Definitions: An error occurred during the evaluation of the S7 PROFIBUS configuration for the specified slave. This is further specified in alarm parameter %2.
%2 = PRM header: the PRM telegram for this slave could not clearly be interpreted.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Check and correct the S7 PROFIBUS configuration.
Program Continuation: Switch control OFF - ON.

27240 PROFIsafe: DP M not running up, DP info: %1

Parameters: %1 = Current information from the DP interface NCK-PLC
Definitions: There is no DP configuration available to the NCK after the time specified via the MD $MN_PLC_RUNNINGUP_TIMEOUT.
Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy:
- Increase MD $MN_PLC_RUNNINGUP_TIMEOUT
- Check the PLC operating status.
- Check the PLC operating system software version.
- Delete the F parameterization in the NCK MD.

Program Continuation: Switch control OFF - ON.

27241 PROFIsafe: DP M version different, NCK: %1, PLC: %2

Parameters: %1 = DP interface version of the NCK
%2 = DP interface version of the PLC
Definitions: The NCK and PLC components have different implementations of the DP interface. The F communication cannot be initialized.
Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check the PLC operating system and NCK software versions. Upgrade the PLC operating system. Delete the NCK F parameterization.
Program Continuation: Switch control OFF - ON.

27242 PROFIsafe: F module %1, %2 faulty

Parameters: %1 = PROFIsafe address
%2 = Incorrect F parameter
Definitions: An error was detected during the evaluation of the F parameters.
%2 = CRC1: CRC specified by F parameters faulty.
%2 = F_WD_Timeout: The monitoring time parameterized in Step 7 is too small for the PROFIsafe cycle time defined by the MD $MN_PROFISAFE_IPO_TIME_RATIO.
%2 = CRC2_Len: CRC message length faulty.
%2 = F_Data_Len: the telegram length defined for the specified module is incorrect.
Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: %2 = CRC1: PLC overall reset, reload the S7 F configuration.
%2 = F_WD_Timeout: reparameterize the PROFIsafe cycle time or F monitoring time.
%2 = CRC2_Len: PLC overall reset, reload the S7 F configuration.
%2 = F_Data_Len: PLC overall reset, reload the S7 F configuration.
Program Continuation: Switch control OFF - ON.

27250 PROFIsafe: configuration in DP M changed; error code %1 - %2

Parameters: %1 = NCK project number
%2 = Current PLC project number
Definitions: The DP master shows a modified S7 PROFIBUS configuration. Error-free operation can no longer be guaranteed. Communication with the F slaves is terminated. Stop D/E is triggered.
Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Restart the PLC/NCK.
Program Continuation: Switch control OFF - ON.

27251 PROFIsafe: F module %1, %2 reports error %3

Parameters: %1 = PROFIsafe address
%2 = Reporting component (master/slave)
%3 = Error code
Definitions: An error occurred in the PROFIsafe communication between the F master and the specified F module which was detected by the reporting component (master/slave). Stop D/E is triggered.
The error code specifies the error type:
- %3 = TO: The parameterized communication timeout was exceeded
- %3 = CRC: A CRC error was detected
- %3 = CN: An error in the time sequence of the F messages was detected
- %3 = SF: F master error, NCK/PLC are no longer synchronous
- %3 = EA: Communication error, slave sends empty messages

Reactions: - Mode group not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Restart F slave modules. Restart the NCK/PLC.
Program Continuation: Clear alarm with the RESET key. Restart part program

27252 PROFIsafe: Slave %1, sign-of-life error

Parameters: %1 = DP slave address
Definitions: The specified DP slave no longer communicates with the master. Stop D/E is triggered.
Reactions: - Mode group not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Restart F slave modules. Restart the NCK/PLC.
Program Continuation: Clear alarm with the RESET key. Restart part program

27253 PROFIsafe: communication fault F master component %1, error %2

Parameters: %1 = Error component (NCK/PLC)
%2 = Error code
Definitions: The F master signals that the communication between the NCK and PLC is no longer working.
The error code %1 specifies the cause:
- %1 = NCK: Link between PROFIsafe and SPL interface is interrupted.
- %1 = PLC: the PLC does no longer execute the OB40 request.
- %1 = PLC-DPM: DP master is no longer in OPERATE status.

Parameter %2 provides further information about the error's cause:
- %2 = 0: NCK-internal sequence error (see %1=NCK).
- %2 = 1,2,4: PLC processing of the OB40 not finished.

Reactions: - Mode group not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Extend the PROFIsafe cycle time via MD $MN_PROFISAFE_IPO_TIME_RATIO.
Program Continuation: Clear alarm with the RESET key. Restart part program

27254 PROFIsafe: F module %1, error on channel %2; %3<ALSI>

Parameters: %1 = PROFIsafe address
%2 = Channel number
%3 = System variable array index extension
Definitions: The F module signals that an error occurred in the interface of the specified channel.
This alarm is only triggered for ET200S F modules.
%2=0: Special meaning: A general error occurred in the F module.
A specific alarm message can be programmed for each of the system variables on the MMC via parameter %3:
- %3 = 1....64: Error in system variables $A_INSE[1...64]
- %3 = 65...128: Error in system variables $A_OUTSE[1...64]

Reactions: - Mode group not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Check wiring. Wiring OK: replace F module.
Program Continuation: Clear alarm with the RESET key. Restart part program

27255 PROFIsafe: F module %1, general error

Parameters: %1 = PROFIsafe address
Definitions: The specified PROFIsafe module signals an error. A more exact specification of the error's cause cannot be made without further assistance.
This alarm is triggered for all types of PROFIsafe slaves.
With ET200S F modules, this error can only occur if there already is a channel error when the cyclical communication between the F master and module is begun.

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Check wiring.

Program Continuation: Clear alarm with the RESET key. Restart part program

27256 PROFIsafe: Current cycle time %1 [ms] > parameterized cycle time

Parameters: %1 = Current PROFIsafe communication cycle time

Definitions: The current PROFIsafe communication cycle time is greater than the value set via MD $MN_PROFISAFE_IPO_TIME_RATIO. The parameterized PROFIsafe communication cycle time is continually exceeded on the PLC side.

Reactions: - Mode group not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Adapt cycle time via MD $MN_PROFISAFE_IPO_TIME_RATIO.

The displayed value, at least, has to be set.

The set cycle time affects the runtime load of the PLC module. This also has to be taken into consideration when making the setting.

Program Continuation: Clear alarm with the RESET key. Restart part program

27299 PROFIsafe: Diagnosis %1 %2 %3 %4

Parameters: %1 = Error code 1

%2 = Error code 2

%3 = Error code 3

%4 = Error code 4

Definitions: Internal error in the NCK PROFIsafe implementation.

Reactions: - Alarm display.

Remedy: Make a note of the error text and contact Siemens A&D MC, Hotline

- Tel 0180 / 5050 - 222 (Germany)
- Fax 0180 / 5050 - 223
- Tel +49-180 / 5050 - 222 (International)
- Fax +49-180 / 5050 - 223
- email techsupport@ad.siemens.de

Program Continuation: Clear alarm with the Delete key or NC START.

28000 NCU link connection to all other NCUs of the link network has been aborted

Definitions: All NCUs in the NCU link network exchange data cyclically (sign-of-life). If this alarm occurs, sign-of-life signals have not been received from any other NCUs on the NCU network. This fault in the link can have various causes:

- Defective hardware.
- The machine data which configure the NCU link are not the same on all NCUs.
- An identical interpolator cycle time has not been selected on all NCUs.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check the IPO cycle on all the NCUs.
If necessary, check NCU link-specific alarms first.

Program Continuation: Switch control OFF - ON.

28001 NCU link connection to the NCU %1 of the link network has been aborted

Parameters: %1 = NCU number

Definitions: All NCUs in the NCU link network exchange data cyclically (sign-of-life). If this alarm occurs, sign-of-life signals have not been received from one other NCU on the NCU network. (see alarm parameters) This fault in the link can have various causes:
- Defective hardware.
- The machine data which configure the NCU link are not identical on all NCUs.
- An identical interpolator cycle time has not been selected on all NCUs.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy:
- Check the IPO cycle on all the NCUs.
- If necessary, check NCU link-specific alarms first.

Program Continuation: Switch control OFF - ON.

28002 Error on activation of machine data, NCU network-wide machine data were modified by NCU %1

Parameters: %1 = NCU number

Definitions: During the activation of machine data with NEWCONFIG or during an operator panel RESET, NCU network-wide machine data were modified on another NCU. This alarm can only occur when a link connection is active.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Repeat the operator action or, if NEWCONFIG is activated by an NC program, terminate the program with Reset.

Program Continuation: Clear alarm with the RESET key. Restart part program

28004 NCU link: NCU %1 of the link network is not on the bus

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. When the NCU link was powered up, the local NCU (indicated by the alarm) detected that the NCU with the number in the alarm parameter was not on the bus although it should be connected according to the MD settings.
This fault in the link can have various causes:
- Defective hardware.
- The machine data which configure the NCU link are not identical on all NCUs.
- An identical interpolator cycle time has not been selected on all NCUs.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check the machine data configuration and link hardware.

Program Continuation: Switch control OFF - ON.

28005 NCU link: NCU %1 of the link network not running synchronously

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. When the NCU link was powered up, the local NCU (indicated by the alarm) detected that the NCU with the number in the alarm parameter was not running synchronously.

This fault in the link can have various causes:

- The machine data which configure the NCU link are not identical on all NCUs.
- An identical interpolator cycle time has not been selected on all NCUs.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data configuration.

Program Continuation: Switch control OFF - ON.

28007 NCU link: conflict in configuration data of NCU %1

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. When the NCU link was powered up, the local NCU (indicated by the alarm) detected a conflict between its configuration and the configuration of the NCU in the alarm parameter.

Example: Machine data LINK_NUM_OF_MODULES defines the number of nodes on the NCU link network. The alarm occurs if this MD has a different setting on different NCUs.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data configuration.

Program Continuation: Switch control OFF - ON.

28008 NCU link: conflict in timer setting of NCU %1

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. When the NCU link was powered up, the local NCU (indicated by the alarm) detected a conflict between its timer configuration and the configuration of the NCU in the alarm parameter.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data configuration.

Program Continuation: Switch control OFF - ON.

28009 NCU link: conflict in bus parameters of NCU %1

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. When the NCU link was powered up, the local NCU (indicated by the alarm) detected a conflict between its timer bus configuration and the configuration of the NCU in the alarm parameter.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check machine data configuration.

Program Continuation: Switch control OFF - ON.

28010 NCU link: the NCU %1 has not received a message

Parameters: %1 = NCU number

Definitions: Error message of the NCU link module. During operation of the NCU link, a message from the local NCU to the NCU specified in the alarm parameter has failed. A hardware error may have occurred (e.g. sporadic disturbances on the communication line).

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: The message does not fail until several attempts have been made to repeat the communication. The number of repetitions can be increased with MD LINK_MAX_RETRY_CTR.

Program Continuation: Switch control OFF - ON.

28011 IPO time insufficient for NCU link. Link cycle time: %1

Parameters: %1 = Microseconds

Definitions: Error message of the NCU link module. All messages must be transmitted within one interpolator cycle. This applies particularly to message retries. The time was not sufficient! The parameter indicates how many microseconds the NCU link module needs in order to send the message.

Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Increase the interpolator cycle time, i.e. modify one of the following MDs on all NCUs.
IPO_SYSCLOCK_TIME_RATIO
SYSCLOCK_CYCLE_TIME
Program Continuation: Switch control OFF - ON.

28012 NCU link: synchronization cycle signal failure %1 times

Parameters: %1 = Number of cycles
Definitions: Error message of the NCU link module that does not occur at NCU 1. The NCU's are synchronized via their own NCU-link clock line. A large number of cycle signals are missing. The parameter indicates how many cycles have failed.
Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Check the hardware.
Program Continuation: Switch control OFF - ON.

28020 NCU link: too many link axes configured %1

Parameters: %1 = Number of link axis connections
Definitions: Unfortunately, the communication capacity of the NCU link is insufficient for this link axis configuration.
The link axis configuration is determined by the following MDs:
• $MN_AXCONF_LOGIC_MACHAX_TAB
• $MN_AXCT_AXCONF_ASSIGN_TAB1 ... and all further container def.
Reactions: - Alarm display.
- Interface signals are set.
- NC not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Connect a smaller number of axes across the link or combine the axes in fewer containers.
Machine data to be changed:
• $MN_AXCONF_LOGIC_MACHAX_TAB
• $MN_AXCT_AXCONF_ASSIGN_TAB1 ... and all further container def.
Program Continuation: Switch control OFF - ON.

28030 Serious alarm on NCU %1, axes in follow-up mode

Parameters: %1 = NCU number
Definitions: All axes are trailing because of a serious alarm on another NCU.
Reactions: - Alarm display.
- NC not ready.
- Mode group not ready, also effective for single axes
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Acknowledge the alarm on the NCU.
Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

28031 Serious alarm on NCU %1 not yet acknowledged, axes still in follow-up mode

Parameters: %1 = NCU number

Definitions: A serious alarm was not yet acknowledged on another NCU. Consequently, all the axes continue to trail.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Acknowledge the alarm on the NCU.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

28032 Emergency stop activated on NCU %1, axes in follow-up mode

Parameters: %1 = NCU number

Definitions: The emergency stop request is active at the PLC-NCK interface on one NCU of the NCU network. Consequently, all axes are trailing.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Remedy the cause of the emergency stop on the NCU and acknowledge the emergency stop via the PLC-NCK interface.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

28033 Emergency stop on NCU % 1, axes still in follow-up mode

Parameters: %1 = NCU number

Definitions: The emergency stop request is active at the PLC-NCK interface on one NCU of the NCU network. Consequently, all axes are trailing.

Reactions:
- Alarm display.
- Interface signals are set.
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- NC Stop on alarm.

Remedy: Remedy the cause of the emergency stop on the NCU and acknowledge the emergency stop via the PLC-NCK interface.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

29033 Channel %1 axis change of axis %2 not possible, PLC axis movement not yet completed

Parameters: %1 = Channel number
%2 = Axis

Definitions: A PLC axis has not yet reached its end position and cannot be returned to a channel or neutralized. This alarm should not occur when PLC data block FC18 is used.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- NC Stop on alarm.

© Siemens AG, 2002. All rights reserved

Remedy: Wait until the axis has reached the end position or terminate the movement with delete distance to go.
Program Continuation: Clear alarm with the RESET key. Restart part program

60000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
Remedy: -
Program Continuation: Clear alarm with the RESET key. Restart part program

61000 Channel %1 block %2 no tool compensation active

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: LONGHOLE, SLOT1, SLOT2, POCKET1 to POCKET4, CYCLE71, CYCLE72, CYCLE90, CYCLE93 to CYCLE96.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: D-correction must be programmed before the cycle call.
Program Continuation: Clear alarm with the RESET key. Restart part program

61001 Channel %1 block %2 thread lead incorrect

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840, CYCLE96, CYCLE97.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Check parameter for the thread size or setting for the lead (contradict each other).
Program Continuation: Clear alarm with the RESET key. Restart part program

61002 Channel %1 block %2 machining type wrongly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The value of the VARI parameter for the machining has been incorrectly specified. Alarm triggered by following cycles: SLOT1, SLOT2, POCKET1 to POCKET4, CYCLE71, CYCLE72, CYCLE76, CYCLE77, CYCLE93, CYCLE95, CYCLE97, CYCLE98.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Modify VARI parameter.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61003 Channel %1 block %2 no feed programmed in the cycle

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The parameter for the feed has been incorrectly specified. Alarm triggered by following cycles: CYCLE71, CYCLE72.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Modify feed parameter.
Program Continuation: Clear alarm with the RESET key. Restart part program

61004 Channel %1 block %2 configuration geometry axis incorrect

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The geometry-axes sequence is wrong. CYCLE328
Reactions: - -
Remedy: -
Program Continuation: Internal

61005 Channel %1 block %2 3rd geometry axis not present

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: With an application on the lathe with no Y-axis in the G18 plane. Alarm triggered by following cycle: CYCLE86.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: -
Program Continuation: Clear alarm with the RESET key. Restart part program

61006 Channel %1 block %2 tool radius too large

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The tool radius is too large for machining. Alarm triggered by following cycles: CYCLE930, CYCLE951, E_CP_CE, E_CP_CO, E_CP_DR, E_PO_CIR, E_PO_REC, F_CP_CE, F_CP_CO, F_CP_DR, F_PO_CIR, F_PO_REC.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Select a smaller tool.
Program Continuation: Clear alarm with the RESET key. Restart part program

61007 Channel %1 block %2 tool radius too small

Parameters: %1 = Channel number
%2 = Block number, label

© Siemens AG, 2002. All rights reserved

Definitions: The tool radius is too small for machining. Alarm triggered by following cycles: CYCLE92, E_CP_CO, E_SL_CIR, F_CP_CO, F_PARTOF, F_SL_CIR.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Select a larger tool.
Program Continuation: Clear alarm with the RESET key. Restart part program

61009 Channel %1 block %2 active tool number = 0

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No tool (T) has been programmed before the cycle call. Alarm triggered by following cycles: CYCLE71, CYCLE72.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Program tool (T).
Program Continuation: Clear alarm with the RESET key. Restart part program

61010 Channel %1 block %2 finishing allowance too large

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The finishing allowance for the base is greater than the total depth. Alarm triggered by following cycle: CYCLE72.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Reduce finishing allowance.
Program Continuation: Clear alarm with the RESET key. Restart part program

61011 Channel %1 block %2 scaling not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A scale factor is active which is illegal for this cycle. Alarm triggered by following cycles: CYCLE71, CYCLE72.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Modify scale factor.
Program Continuation: Clear alarm with the RESET key. Restart part program

61012 Channel %1 block %2 different scaling on the plane

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61013 Channel %1 block %2 basic settings were changed, program cannot be executed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The basic settings are not compatible with the generated program. Alarm triggered by following cycles: E_CP_CE, E_CP_CO, E_CP_DR, F_CP_CE, F_CP_CO, F_CP_DR.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check and, if necessary, change the basic settings.

Program Continuation: Clear alarm with the RESET key. Restart part program

61101 Channel %1 block %2 reference plane incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE71, CYCLE72, CYCLE81 to CYCLE90, CYCLE840, SLOT1, SLOT2, POCKET1 to POCKET4, LONGHOLE.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: With the relative specification of the depth, either different values for the reference plane and the retraction plane must be selected or an absolute value must be specified for the depth.

Program Continuation: Clear alarm with the RESET key. Restart part program

61102 Channel %1 block %2 no spindle direction programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE86, CYCLE87, CYCLE88, CYCLE840, POCKET3, POCKET4.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Parameter SDIR (or SDR in CYCLE840) must be programmed.

Program Continuation: Clear alarm with the RESET key. Restart part program

61103 Channel %1 block %2 number of drillings is zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No value for the number of holes has been programmed. Alarm triggered by following cycles: HOLES1, HOLES2.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61104 Channel %1 block %2 contour violation of slots/long holes

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Incorrect parameterization of the milling pattern in the parameters which define the position of the slots/elongated holes on the circle and their form. Alarm triggered by following cycles: SLOT1, SLOT2, LONGHOLE.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61105 Channel %1 block %2 cutter radius too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The diameter of the cutter used is too large for the form to be machined. Alarm triggered by following cycles: SLOT1, SLOT2, POCKET1 to POCKET4, LONGHOLE, CYCLE90.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Either a tool with a smaller radius has to be used or the contour must be modified.

Program Continuation: Clear alarm with the RESET key. Restart part program

61106 Channel %1 block %2 number or distance of circular elements

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Incorrect parameterization of NUM or INDA. The layout of the circle elements within a full circle is not possible. Alarm triggered by following cycles: HOLES2, LONGHOLE, SLOT1, SLOT2.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Correct parameterization.

Program Continuation: Clear alarm with the RESET key. Restart part program

61107 Channel %1 block %2 first drilling depth incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: First drilling depth is in the opposite direction to the total drilling depth. Alarm triggered by following cycle: CYCLE83.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Modify drilling depth.

Program Continuation: Clear alarm with the RESET key. Restart part program

61108 Channel %1 block %2 illegal values for parameters _RAD1 and _DP1

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The parameters _RAD1 and _DP for defining the path for the depth infeed have been incorrectly specified. Alarm triggered by following cycles: POCKET3, POCKET4.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Modify parameter.

Program Continuation: Clear alarm with the RESET key. Restart part program

61109 Channel %1 block %2 parameter _CDIR incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The value of the parameter for the cutting direction _CDIR has been incorrectly defined. Alarm triggered by following cycles: POCKET3, POCKET4.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Modify parameter.

Program Continuation: Clear alarm with the RESET key. Restart part program

61110 Channel %1 block %2 finishing allowance at the base > depth infeed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The finishing allowance at the base has been specified greater than the maximum depth infeed. Alarm triggered by following cycles: POCKET3, POCKET4.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Either reduce finishing allowance or increase depth infeed.

Program Continuation: Clear alarm with the RESET key. Restart part program

61111 Channel %1 block %2 infeed width > tool diameter

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed infeed width is greater than the diameter of the active tool. Alarm triggered by following cycles: CYCLE71, POCKET3, POCKET4.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Infeed width must be reduced.

Program Continuation: Clear alarm with the RESET key. Restart part program

61112 Channel %1 block %2 tool radius negative

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The radius of the active tool is negative. This is illegal. Alarm triggered by following cycles: CYCLE72, CYCLE76, CYCLE77, CYCLE90.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61113 Channel %1 block %2 parameter _CRAD for corner radius too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The parameter for the corner radius _CRAD has been specified too large. Alarm triggered by following cycle: POCKET3.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Parameter must be reduced.

Program Continuation: Clear alarm with the RESET key. Restart part program

61114 Channel %1 block %2 machining direction G41/G42 incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The machining direction of the cutter radius compensation G41/G42 has been incorrectly selected. Alarm triggered by following cycle: CYCLE72.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Change machining direction.

Program Continuation: Clear alarm with the RESET key. Restart part program

61115 Channel %1 block %2 approach or retract mode (straight line/circle/plane/space) incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The approach or retract mode to/from the contour has been incorrectly defined. Alarm triggered by following cycle: CYCLE72.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Check parameter _AS1 or _AS2.

Program Continuation: Clear alarm with the RESET key. Restart part program

61116 Channel %1 block %2 approach or retraction path = 0

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The approach or retract path has been specified with zero. Alarm triggered by following cycle: CYCLE72.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Check parameter _LP1 or _LP2.

Program Continuation: Clear alarm with the RESET key. Restart part program

61117 Channel %1 block %2 active tool radius <= 0

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The radius of the active tool is negative or zero. Alarm triggered by following cycles: CYCLE71, POCKET3, POCKET4.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Modify radius.

Program Continuation: Clear alarm with the RESET key. Restart part program

61118 Channel %1 block %2 length or width = 0

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The length or width of the milling area is illegal. Alarm triggered by following cycle: CYCLE71.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Check parameters _LENG and _WID.

Program Continuation: Clear alarm with the RESET key. Restart part program

61119 Channel %1 block %2 nominal or core diameter incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The nominal or core diameter was incorrectly programmed. Alarm triggered by following cycles: CYCLE70, E_MI_TR, F_MI_TR.

© Siemens AG, 2002. All rights reserved

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check thread geometry.

Program Continuation: Clear alarm with the RESET key. Restart part program

61120 Channel %1 block %2 internal/external thread type not defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The thread type (internal/external) was not defined. Alarm triggered by following cycles: CYCLE70.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: The internal/external thread type must be entered.

Program Continuation: Clear alarm with the RESET key. Restart part program

61121 Channel %1 block %2 number of teeth per cutting edge missing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No value was entered for the number of teeth per cutting edge. Alarm triggered by following cycles: CYCLE70.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Enter the number of teeth/cutting edges for the active tool into the tool list.

Program Continuation: Clear alarm with the RESET key. Restart part program

61122 Channel %1 block %2 safety clearance on the plane incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The safety clearance is negative or zero. This is not allowed.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Define the safety clearance.

Program Continuation: Clear alarm with the RESET key. Restart part program

61124 Channel %1 block %2 infeed width is not programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE71.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

© Siemens AG, 2002. All rights reserved

Remedy: With active simulation without tool, a value for the infeed width _MIDA must always be programmed.
Program Continuation: Clear alarm with the RESET key. Restart part program

61125 Channel %1 block %2 technology selection in parameter _TECHNO incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Check parameter _TECHNO.
Program Continuation: Clear alarm with the RESET key. Restart part program

61126 Channel %1 block %2 thread length too short

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE840.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Program lower spindle speed/raise reference plane.
Program Continuation: Clear alarm with the RESET key. Restart part program

61127 Channel %1 block %2 transmission ratio of tapping axis incorrectly defined (machine data)

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Check machine data 31050 and 31060 in the appropriate gear stage of the drilling axis.
Program Continuation: Clear alarm with the RESET key. Restart part program

61128 Channel %1 block %2 dipping angle = 0 for dipping with oscillation or helix

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: SLOT1.
Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop
Remedy: Check parameter _STA2.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61180 Channel %1 block %2 no name assigned to swivel data block, although MD $MN_MM_NUM_TOOL_CARRIER > 1

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Although there are several swivel data blocks, no unique names were assigned. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Assign unique names for swivel data blocks.

Program Continuation: Clear alarm with the RESET key. Restart part program

61181 Channel %1 block %2 NCK software version unsufficient (missing TOOLCARRIER functionality)

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Swivelling is not possible with the current NCK software version. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Upgrade NCK software.

Program Continuation: Clear alarm with the RESET key. Restart part program

61182 Channel %1 block %2 name of swivel data block unknown

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: the specified name of the swivel data block is unknown. Alarm triggered by following cycles: CYCLE800, E_TCARR.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check the name of the swivel data block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61183 Channel %1 block %2 retraction mode GUD7 _TC_FR beyond value range 0... 2

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The retraction mode value lies outside of the valid range. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61184 Channel %1 block %2 no solution possible with current input angle values

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The surface defined via the input angle cannot be processed with the machine. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check the angle entered for the swivelling of the machining plane.

Program Continuation: Clear alarm with the RESET key. Restart part program

61185 Channel %1 block %2 no or incorrect (min > max) rotary axis angle ranges assigned

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The rotary axis angle range is invalid. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.

Program Continuation: Clear alarm with the RESET key. Restart part program

61186 Channel %1 block %2 rotary axis vectors invalid --> Check installation and start-up of the swivel cycle CYCLE800

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A rotary axis vector is invalid. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.

Program Continuation: Clear alarm with the RESET key. Restart part program

61188 Channel %1 block %2 no axis name for the 1st axis assigned -> Check installation and start-up of the swivel cycle CYCLE800

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No axis name was specified for the 1st rotary axis. Alarm triggered by following cycles: CYCLE800.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61200 Channel %1 block %2 too many elements in the machining block

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61201 Channel %1 block %2 wrong sequence in the machining block

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The sequence of elements in the machining block is invalid. Alarm triggered by following cycles: CYCLE108, E_CP_CE, E_CP_DR, E_MANAGE, F_CP_CE, F_CP_DR, F_MANAGE.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Sort the sequence in the machining block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61202 Channel %1 block %2 no technology cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No technology cycle was programmed in the machining block. Alarm triggered by following cycles: CYCLE108, E_MANAGE, F_MANAGE.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program a technology block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61203 Channel %1 block %2 no positioning cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No positioning cycle was programmed in the machining block. Alarm triggered by following cycles: CYCLE108, E_MANAGE, F_MANAGE.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program positioning block.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61204 Channel %1 block %2 unknown technology cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The specified technology cycle in the machining block is unknown. Alarm triggered by following cycles: E_MANAGE, F_MANAGE.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Delete and reprogram the technology block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61205 Channel %1 block %2 unknown positioning cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The specified positioning cycle in the machining block is unknown. Alarm triggered by following cycles: E_MANAGE, F_MANAGE.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Delete and reprogram the positioning block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61210 Channel %1 block %2 block search element not found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The element specified for the block search does not exist. Alarm triggered by following cycles: E_MANAGE, E_PS_CIR, E_PS_MRX, E_PS_SEQ, F_MANAGE, F_PS_CIR, F_PS_MRX, F_PS_SEQ.

Reactions:
- Interpreter stop
- Channel not ready.
- Interface signals are set.
- Alarm display.

Remedy: Repeat block search.

Program Continuation: Clear alarm with the RESET key. Restart part program

61211 Channel %1 block %2 absolute reference missing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61212 Channel %1 block %2 wrong tool type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool type is not suitable for machining. Alarm triggered by following cycles: CYCLE92, CYCLE951, E_DR, E_DR_PEC, E_DR_SIN, F_DR, F_DRILL, F_DRILLC, F_DRILLD, F_DR_PEC, F_DR_SIN, F_GROOV, F_MT_LEN, F_PARTOF, F_ROUGH, F_ROU_Z, F_SP_EF, F_TAP, F_TR_CON, F_UCUT_T.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Select a new tool type.

Program Continuation: Clear alarm with the RESET key. Restart part program

61213 Channel %1 block %2 circle radius too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE77.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61214 Channel %1 block %2 no lead programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No thread lead was entered. Alarm triggered by following cycles: E_CR_HEL, E_PO_CIR, E_PO_REC, F_PO_CIR, F_PO_REC.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program a lead.

Program Continuation: Clear alarm with the RESET key. Restart part program

61215 Channel %1 block %2 raw dimension incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE76, CYCLE77.

Reactions:
- Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61216 Channel %1 block %2 feed/tooth only possible with milling tools

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Feed per tooth is only possible with milling tools. Alarm triggered by following cycles: E_TFS, F_TFS.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: As alternative, set a different feed type.

Program Continuation: Clear alarm with the RESET key. Restart part program

61217 Channel %1 block %2 cutting speed for tool radius 0 programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: To be able to work with cutting speed, the tool radius has to be specified. Alarm triggered by following cycles: E_TFS, E_DR_TAP, F_TFS, F_DR_TAP.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Enter a value for cutting speed.

Program Continuation: Clear alarm with the RESET key. Restart part program

61218 Channel %1 block %2 feed/tooth programmed, but number of teeth is zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: For feed per tooth, the number of teeth has to be specified. Alarm triggered by following cycles: E_TFS, E_DR_BGF, F_TFS.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Enter the number of teeth on the milling tool in the "Tool list" menu.

Program Continuation: Clear alarm with the RESET key. Restart part program

61222 Channel %1 block %2 plane infeed greater than the tool diameter

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The plane infeed must not be greater than the tool diameter. Alarm triggered by following cycles: CYCLE79, E_PO_CIR, E_PO_REC, F_PO_CIR, F_PO_REC.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Reduce plane infeed.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61223 Channel %1 block %2 approach path too short

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The approach path must not be less than zero. Alarm triggered by following cycles: E_MI_CON, F_MI_CON.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Enter a greater value for the approach path.

Program Continuation: Clear alarm with the RESET key. Restart part program

61224 Channel %1 block %2 retract path too short

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The retract path must not be less than zero. Alarm triggered by following cycles: E_MI_CON, F_MI_CON.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Enter a greater value for the retract path.

Program Continuation: Clear alarm with the RESET key. Restart part program

61225 Channel %1 block %2 swivel data block unknown

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An attempt was made to access a swivel data block which has not been defined.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Select another swivel data block or define a new swivel data block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61226 Channel %1 block %2 swivel head cannot be exchanged

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The parameter "Swivel data block" is set to "No". In spite of this, an attempt has been made to change the swivel head.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Set the parameter "Swivel data block" in the start-up screen form "Rotary axes" to "Automatic" or "Manual".

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61230 Channel %1 block %2 tool probe diameter too small

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The tool probe is not correctly calibrated.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Calibrate the tool probe.
Program Continuation: Clear alarm with the RESET key. Restart part program

61231 Channel %1 block %2 ShopMill program %3 cannot be executed, as it has not been tested by ShopMill

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name
Definitions: Before a ShopMill program can be executed, it has to be tested by ShopMill. Alarm triggered by following cycles: E_HEAD.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: The program has to be simulated first in ShopMill or loaded into the operating mode "Machine auto" by ShopMill.
Program Continuation: Clear alarm with the RESET key. Restart part program

61232 Channel %1 block %2 loading of magazine tool not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Only manual tools may be loaded into a swivel head in which the tools can only be manually loaded.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Load a manual tool into the swivel head or set the parameter "Tool change" in the start-up screen form "Rotary axes" to "Automatic".
Program Continuation: Clear alarm with the RESET key. Restart part program

61233 Channel %1 block %2: Thread inclination incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Check thread geometry.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61234

Channel %1 block %2 ShopMill subroutine %4 cannot be executed, as it has not been tested by ShopMill

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Before a ShopMill subroutine can be used, it has to be tested by ShopMill. Alarm triggered by following cycles: E_HEAD.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: The subroutine has to be simulated first in ShopMill or loaded into the ShopMill operating mode "Machine auto".

Program Continuation: Clear alarm with the RESET key. Restart part program

61235

Channel %1 block %2: ShopTurn program %4 cannot be executed, as it has not been tested by ShopTurn.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name

Definitions: Before a ShopTurn program can be executed, it has to be tested by ShopTurn.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Simulate the subroutine first in ShopTurn or load it into the ShopTurn operating mode "Machine auto".

Program Continuation: Clear alarm with the RESET key. Restart part program

61236

Channel %1 block %2: ShopTurn subroutine %4 cannot be executed, as it has not been tested by ShopTurn.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Subroutine name

Definitions: Before a ShopTurn program can be executed, it has to be tested by ShopTurn.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Simulate the subroutine first in ShopTurn or load it into the ShopTurn operating mode "Machine auto".

Program Continuation: Clear alarm with the RESET key. Restart part program

61237

Channel %1 block %2: Retraction direction unknown. Manually retract tool!

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: -

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Manually retract the tool from the retraction area defined in the program header and restart the program.
Program Continuation: Internal

61238 Channel %1 block %2: Machining direction unknown.

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Please contact the responsible Siemens regional office.
Program Continuation: Clear alarm with the RESET key. Restart part program

61239 Channel %1 block %2: Tool change point lies in the retraction area!

Definitions: The tool change point has to be far enough outside the retraction area so that when the revolver is swivelled, no tool extends into the retraction area.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Specify another tool change point.
Program Continuation: Clear alarm with the RESET key. Restart part program

61240 Channel %1 block %2: Wrong feed type

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Check feed type.
Program Continuation: Clear alarm with the RESET key. Restart part program

61241 Channel %1 block %2: No retraction plane defined for this machining direction.

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Define more retraction planes.
Program Continuation: Clear alarm with the RESET key. Restart part program

61242 Channel %1 block %2: Wrong machining direction

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Check programmed parameters.
Program Continuation: Clear alarm with the RESET key. Restart part program

61243 Channel %1 block %2: Correct tool change point, tool tip is in retraction area!

Definitions: The tool change point has to be far enough outside the retraction area so that when the revolver is swivelled, no tool extends into the retraction area.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Specify another tool change point.
Program Continuation: Clear alarm with the RESET key. Restart part program

61244 Channel %1 block %2: Thread lead change results in an undefined thread

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check thread geometry.
Program Continuation: Clear alarm with the RESET key. Restart part program

61246 Channel %1 block %2: Safety clearance too small

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Increase safety clearance.
Program Continuation: Internal

61247 Channel %1 block %2: Blank radius too small

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Increase blank radius.
Program Continuation: Clear alarm with the RESET key. Restart part program

61248 Channel %1 block %2: Infeed too small

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Increase infeed.
Program Continuation: Clear alarm with the RESET key. Restart part program

61249 Channel %1 block %2: Number of edges too small

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Increase number of edges.
Program Continuation: Clear alarm with the RESET key. Restart part program

61250 Channel %1 block %2: Key width/Edge length too small

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Increase key width/edge length.
Program Continuation: Clear alarm with the RESET key. Restart part program

61251 Channel %1 block %2: Key width/Edge length too large

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Decrease key width/edge length.
Program Continuation: Clear alarm with the RESET key. Restart part program

61252 Channel %1 block %2: Chamfer/Radius too large

Definitions: -
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Decrease chamfer/radius.
Program Continuation: Clear alarm with the RESET key. Restart part program

61253 Channel %1 block %2: No finishing allowance programmed

Definitions: No finishing allowance was entered.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program a finishing allowance.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61254 Channel %1 block %2: Error when traversing to fixed stop

Definitions: -

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: specify another Z1 position for gripping the counterspindle.

Program Continuation: Internal

61255 Channel %1 block %2: Cut-off error: Tool breakage?

Definitions: Cut-off could not be completely carried out.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Check the tool.

Program Continuation: Clear alarm with the RESET key. Restart part program

61301 Channel %1 block %2 measuring probe does not switch

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The measuring distance was completely traversed but no switching signal was generated at the measuring input specified by MD 9750or MD 9751.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy:
- Check measuring probe connection
- Set a longer measuring distance via MD 9752, 9753, 9754, 9755
- When measuring an edge: Position closer to the edge
- For spigots/holes: Position roughly over the middle
- Check value for spigot/hole diameter

Program Continuation: Clear alarm with the RESET key. Restart part program

61302 Channel %1 block %2 measuring probe collision

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The measuring probe collided with an obstacle when being positioned.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy:
- Check spigot diameter (may be too small) check measuring distance (may be to long)

Program Continuation: Clear alarm with the RESET key. Restart part program

61303 Channel %1 block %2 safe area exceeded

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Measuring result deviates greatly from specified value for the spigot/hole diameter.

© Siemens AG, 2002. All rights reserved

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: • Check radius or diameter. Check measuring location (e.g. inaccuracy due to filings).

Program Continuation: Clear alarm with the RESET key. Restart part program

61308 Channel %1 block %2 check measuring distance 2a

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: A traversing path for measuring was generated that's size was specified by MD's that describe the maximum distance before and after the switching position (workpiece edge) and that must have a value greater than 0.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: • Enter a measuring distance that equals 0. Check MD 9752, 9753, 9754, 9755.

Program Continuation: Clear alarm with the RESET key. Restart part program

61309 Channel %1 block %2 check measuring probe type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Measuring probe type: 3D-probe inactive.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: The measuring probe has to be of the "3D-probe" type in the tool management.

Program Continuation: Clear alarm with the RESET key. Restart part program

61310 Channel %1 block %2 scale factor is active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Scale factor = scaling is active.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Switch off the active scale factor in the program. Measuring is not possible with an active scale factor.

Program Continuation: Clear alarm with the RESET key. Restart part program

61311 Channel %1 block %2 no D number is active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No tool offset for the measuring probe (for workpiece measurement) or no tool offset for the active tool (for tool measurement) is selected.

© Siemens AG, 2002. All rights reserved

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Select the tool's tool edge number D.

Program Continuation: Clear alarm with the RESET key. Restart part program

61316 Channel %1 block %2 center point and radius cannot be determined.

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No circle can be calculated from the measured points, as all measured points lie on a straight line.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program change

Program Continuation: Clear alarm with the RESET key. Restart part program

61332 Channel %1 block %2 change tool tip position

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool tip is below the measuring probe surface (e.g. for a ring gauge or cube).

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Place the tool above the measuring probe surface.

Program Continuation: Clear alarm with the RESET key. Restart part program

61338 Channel %1 block %2 positioning velocity is zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: For some measuring versions, e.g. measuring spigots, in addition to the actual measuring paths, intermediate paths were generated that are traversed with a specified feed.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Set the appropriate feed (plane feed/infeed) via MD 9757 or 9758.

Program Continuation: Clear alarm with the RESET key. Restart part program

61601 Channel %1 block %2 finished-part diameter too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: An incorrect finished-part diameter has been programmed. Alarm triggered by following cycles: CYCLE94, CYCLE96.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61602 Channel %1 block %2 tool width incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Plunge cutter is larger than the programmed groove width. Alarm triggered by following cycle: CYCLE93.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61603 Channel %1 block %2 groove form incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Radii/chamfers at the groove base do not match the groove width. Face groove on a contour element running parallel to the longitudinal axis is not possible. Alarm triggered by following cycle: CYCLE93.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61604 Channel %1 block %2 active tool violates programmed contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Contour violation in the relief cut elements due to the tool clearance angle of the tool used. Alarm triggered by following cycle: CYCLE95.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Use a different tool or check the contour subroutine.

Program Continuation: Clear alarm with the RESET key. Restart part program

61605 Channel %1 block %2 contour incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Illegal relief cut element detected. Alarm triggered by following cycles: CYCLE76, CYCLE77, CYCLE95.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61606 Channel %1 block %2 contour processing error

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Error detected with the contour preparation This alarm is always issued in connection with an NCK alarm 10930...10934, 15800 or 15810. Alarm triggered by following cycle: CYCLE95.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61607 Channel %1 block %2 starting point incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The starting point reached before the cycle call does not lie outside the rectangle described by the contour subroutine. Alarm triggered by following cycle: CYCLE95.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61608 Channel %1 block %2 wrong cutting edge position programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE94, CYCLE96.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: A cutting edge position 1...4, matching the undercut form, must be programmed.

Program Continuation: Clear alarm with the RESET key. Restart part program

61609 Channel %1 block %2 form incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE94, CYCLE96.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Check parameter for the undercut form.

Program Continuation: Clear alarm with the RESET key. Restart part program

61610 Channel %1 block %2 no infeed depth programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77, CYCLE96.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61611 Channel %1 block %2 no intersection point found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No intersection could be calculated with the contour. Alarm triggered by following cycle: CYCLE95.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Check contour programming or modify infeed depth.

Program Continuation: Clear alarm with the RESET key. Restart part program

61612 Channel %1 block %2 thread axis cutting not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE97, CYCLE98.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: -

Program Continuation: Clear alarm with the RESET key. Restart part program

61613 Channel %1 block %2 undercut position incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE94, CYCLE96.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

© Siemens AG, 2002. All rights reserved

Remedy: Check value in parameter _VARI.
Program Continuation: Clear alarm with the RESET key. Restart part program

61800 Channel %1 block %2: External CNC system missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Machine data for external language MD18800: $MN_MM_EXTERN_LANGUAGE or option bit 19800 $ON_EXTERN_LANGUAGE is not set.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61801 Channel %1 block %2: Wrong G code selected

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the program call CYCLE300<value> an impermissible numerical value was programmed for the entered CNC_System, or in the Cycles_Setting_Datum an incorrect value for the G_Code_System was set.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61802 Channel %1 block %2: Wrong axis type

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed axis is assigned to a spindle
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61803 Channel %1 block %2 programmed axis not present

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed axis is not in the system.
Alarm triggered by following cycles: CYCLE83, CYCLE84, CYCLE840.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Check parameter _AXN.
Check MD20050-20080.
Program Continuation: Clear alarm with the RESET key. Restart part program

61804 Channel %1 block %2: Progr. position exceeds reference point

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed intermediate position or actual position is behind the reference point.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61805 Channel %1 block %2: The value is absolutely and incrementally programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The programmed intermediate position is both absolutely as well as incrementally programmed.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61806 Channel %1 block %2: Incorrect axis assignment

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The axis-assignment sequence is wrong.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61807 Channel %1 block %2 wrong spindle direction programmed (active)

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE840.
The programmed spindle direction contradicts the spindle direction planned for the cycle.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: Check parameters SDR and SDAC.
Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

61808 Channel %1 block %2: The final drilling depth or individual drilling depth is missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The total depth Z or individual drilling depth Q is missing from theG8xblock (initial cycle call).
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61809 Channel %1 block %2: Impermissible drilling position

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

61810 Channel %1 block %2: ISO-G code not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the call block an impermissible ISO axis name was programmed.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61811 Channel %1 block %2: Impermissible ISO axis name

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the call block an impermissible numerical value was programmed.
Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
Remedy: -
Program Continuation: Internal

61812 Channel %1 block %2: Value(s) in the external cycle call incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: In the call block an impermissible numerical value was programmed.

© Siemens AG, 2002. All rights reserved

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: -

Program Continuation: Internal

61813 Channel %1 block %2: GUD value incorrectly defined

Definitions: An impermissible numerical value was entered in the cycles-setting data.

Reactions: - -

Remedy: -

Program Continuation: Internal

61814 Channel %1 block %2: Polar coordinates not possible with cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: -

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: -

Program Continuation: Internal

61815 Channel %1 block %2: G40 not active

Parameters: %1 = Channel number
%2 = Block number

Definitions: G40 was inactive before the cycle call.

Reactions: - Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: -

Program Continuation: Internal

61816 Channel %1 block %2: axes are not at the reference point

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: -

Reactions: - -

Remedy: -

Program Continuation: Internal

61817 Channel %1 block %2: The axis coordinates are within the protection zone

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: -

Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

61818 Channel %1 block %2: The axis area limit values are identical

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

62000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62100 Channel %1 block %2 no drilling cycle active

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No modal drilling cycle has been called before the drilling pattern cycle call. Alarm triggered by following cycles: HOLES1, HOLES2.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62101 Channel %1 block %2: Incorrect cutting direction - G3 is generated

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Synchronous or reverse rotation programmed. But the spindle does not rotate at a cycle call.
Reactions: - Alarm display.
Remedy: Check whether the spindle rotates.
Program Continuation: Clear alarm with the Delete key or NC START.

62103 Channel %1 block %2: The finishing allowance is not programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No finishing allowance is programmed, although it is necessary for this machining.
Reactions: - Alarm display.
Remedy: Program a finishing allowance.
Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

62105 Channel %1 block %2 number of columns or lines is zero

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE801.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62180 Channel %1 block %2 no name assigned to swivel data block although machine data $MN_MM_NUM_TOOL_CARRIER > 1

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62181 Channel %1 block %2 NCK software version unsufficient (missing TOOLCARRIER functionality)

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62182 Channel %1 block %2: Load swivel head

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: No swivel head is active. Alarm triggered by following cycles: E_TCARR, F_TCARR.
Reactions: - Alarm display.
Remedy: Request to load a swivel head.
Program Continuation: Clear alarm with the Delete key or NC START.

62183 Channel %1 block %2 retraction mode GUD7 _TC_FR beyond value range 0...2

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62184 Channel %1 block %2 no solution possible with current input angle values

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62185 Channel %1 block %2 no end stop assigned to rotary axes

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: Check swivel cycle CYCLE800 start-up.
Program Continuation: Clear alarm with the Delete key or NC START.

62186 Channel %1 block %2 illegal rotary axis vectors

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: Check swivel cycle CYCLE800 start-up.
Program Continuation: Clear alarm with the Delete key or NC START.

62187 Channel %1 block %2 name of swivel data block unknown

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE800.
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

62200 Channel %1 block %2: Start spindle

Definitions: -
Reactions: - Alarm display.
Remedy: Start the tool spindle before machining the thread.
Program Continuation: Clear alarm with the RESET key. Restart part program

63000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: -
Reactions: - Alarm display.
Remedy: -
Program Continuation: Clear alarm with the Delete key or NC START.

65000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: References: The current alarm text, the error description and the remedial measures for the user cycle alarms can be found in the Programming Guide of the machine manufacturer.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.

Remedy: Refer to the manual on user cycles.

Program Continuation: Clear alarm with the RESET key. Restart part program

66000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: References: The current alarm text, the error description and the remedial measures for the user cycle alarms can be found in the Programming Guide of the machine manufacturer.

Reactions: - Alarm display.
- Interface signals are set.
- NC Start disable in this channel.
- Interpreter stop

Remedy: Refer to the manual on user cycles.

Program Continuation: Clear alarm with the RESET key. Restart part program

67000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: References: The current alarm text, the error description and the remedial measures for the user cycle alarms can be found in the Programming Guide of the machine manufacturer.

Reactions: - Alarm display.

Remedy: Refer to the manual on user cycles.

Program Continuation: Clear alarm with the Delete key or NC START.

68000 Channel %1 block %2:

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: References: The current alarm text, the error description and the remedial measures for the user cycle alarms can be found in the Programming Guide of the machine manufacturer.

Reactions: - Alarm display.

Remedy: Refer to the manual on user cycles.

Program Continuation: Clear alarm with the Delete key or NC START.

70000 Compile cycle alarm

Definitions: References: The current alarm text, the error description and the remedial measures for the Compile Cycle Alarms can be found in the Manual and the Planning Guide for Compile Cycles.

Reactions: - Alarm display.

Remedy: Refer to the manual on Compile Cycles.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

75000 OEM alarm

Definitions: References: The current alarm text, the error description and the remedial measures for the OEM alarms can be found in the OEM Description.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. Refer to the OEM Description.

Program Continuation: Clear alarm with the Delete key or NC START.

75005 Channel %1 block %2 CLC: General programming error

Parameters: %1 = Channel number

%2 = Block number

Definitions: The activation/deactivation command for the clearance control "CLC(..)" accepts only the values 2, 1, 0 and -1 as call parameters. This alarm signals that parameters are incorrect or missing. The activation command CLC(2) with monitoring of sensor collision signal is accepted only if a valid digital input is configured for the monitoring signal in MD $MC_CLC_SENSOR_TOUCHED_INPUT.

Reactions: - Alarm display.

Remedy: Modify part program. Modify part program. Configure digital input for collision evaluation in MD if necessary.

Program Continuation: Clear alarm with the RESET key. Restart part program

75010 Channel %1 block %2 CLC_LIM value exceeds MD limit

Parameters: %1 = Channel number

%2 = Block number

Definitions: One of the limits for the position offset of the clearance control programmed with CLC_LIM(...,...) is greater than the permissible limitation set in the associated MD.
$MC_CLC_SENSOR_LOWER_LIMIT[1] or
$MC_CLC_SENSOR_UPPER_LIMIT[1].

Reactions: - Alarm display.

Remedy: Modify part program. Raise limitation in appropriate machine data if necessary.

Program Continuation: Clear alarm with the RESET key. Restart part program

75015 Channel %1 block %2 CLC(0) with active TOC

Parameters: %1 = Channel number

%2 = Block number

Definitions: The 3D clearance control has been switched off with CLC(0) while tool radius compensation is still active (G41/G42). Since CLC(0) empties the internal block buffer and transfers the current position offset of the clearance control as a "contour jump" to the interpreter, TRC must be deactivated when this command is issued.

Reactions: - Alarm display.

Remedy: Modify part program: Switch off active G41/G42 before CLC(0) or do not switch of clearance control, but just "freeze" temporarily (CLC_GAIN=0.0) or cancel the position offset mechanically with CLC(-1).

Program Continuation: Clear alarm with the RESET key. Restart part program

75016 Channel %1 block %2 CLC: orientation changed for TRAFOOF

Parameters: %1 = Channel number

%2 = Block number

© Siemens AG, 2002. All rights reserved

Definitions: 1. The 2D/3D clearance control has been switched off before the transformation. The tool direction according to G17/G18/G19 has been applied as the control direction. Switching on the transformation with rotary axis settings that define a different tool orientation requires an orientation step change and is therefore rejected.

2. The transformation has been switched off temporarily (TRAFOOF) while clearance control is still active. When the transformation is switched on again, the tool orientation must be the same as when it was switched off, i.e. the rotary axes must not be moved while the transformation is deactivated.

Reactions: - Alarm display.

Remedy: Modify part program: Do not switch on the clearance control until the transformation is already active or make sure that the required conditions relating to orientation are observed.

Program Continuation: Clear alarm with the RESET key. Restart part program

75020 Channel %1 CLC position offset at lower limit %2

Parameters: %1 = Channel number

%2 = Limit value

Definitions: The position offset generated by the overlaid motion has reached the limit set in MD $MC_CLC_SENSOR_LOWER_LIMIT or programmed with CLC_LIM(...,...).

Reactions: - Alarm display.

Remedy: Check position and form of the workpiece. If necessary, program further limits.

Program Continuation: Clear alarm with the Delete key or NC START.

75021 Channel %1 CLC position offset at upper limit %2

Parameters: %1 = Channel number

%2 = Limit value

Definitions: The position offset generated by the overlaid motion has reached the limit set in MD $MC_CLC_SENSOR_UPPER_LIMIT or programmed with CLC_LIM(...,...).

Reactions: - Alarm display.

Remedy: Depending on setting in bit 1 of MD

$MC_CLC_SPECIAL_FEATURE_MASK:

Bit 1 = 0: Cancel key

Bit 1 = 1: Reset.

Program Continuation: Clear alarm with the Delete key or NC START.

75025 Channel %1 CLC stopped because sensor head has been touched

Parameters: %1 = Channel number

Definitions: The collision monitor of the sensor tip has signaled "Sensor touched".

Reactions: - Alarm display.

Remedy: The part program can be continued with NC start. The overlaid motion then returns to the control distance.

Program Continuation: Clear alarm with the Delete key or NC START.

75050 Channel %1 wrong MD configuration, error code %2

Parameters: %1 = Channel number

%2 = Error code

© Siemens AG, 2002. All rights reserved

Definitions: Incorrect configuration in MD $MA_CC_MASTER_AXIS
%2 = 2 This or the CC_Master axis is a spindle.
%2 = 4 No coupling between rotary and linear axes.
%2 = 8 Axes must not be exchange axes.
Reactions: - Alarm display.
Remedy: Check machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

75051 Channel %1 CC_COPON CC_COPOFF error code %2

Parameters: %1 = Channel number
%2 = Error code
Definitions: %2 = 1 Wrong argument programmed
%2 = 10 An axis which is not involved in a defined coupling has been programmed in CC_COPON(x)..
%2 = 20 Too many arguments
%2 = 100 Internal error
%2 = 200 Internal error
Reactions: - Interpreter stop
Remedy: Modify part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

75060 Channel %1 tolerance window exceeded axis %2

Parameters: %1 = Channel number
%2 = Axis name
Definitions: The actual value difference between the CC_Slave axis %2 and its CC_Master axis is outside the configured tolerance window.
Reactions: - Alarm display.
Remedy: Check configured tolerance window.
Compare dynamic response settings of coupled axes.
Check mechanical components of axes.
Program Continuation: Clear alarm with the RESET key. Restart part program

75061 Channel %1 coupling active axis %2

Parameters: %1 = Channel number
%2 = Axis name
Definitions: Machine data MD 63000: CC_MASTER_AXIS has been changed when the coupling was active.
Reactions: - Alarm display.
Remedy: Reset machine data to its old value, switch off the coupling and then enter the new value.
Program Continuation: Clear alarm with the RESET key. Restart part program

75062 Channel %1 axes not in standstill axis %2

Parameters: %1 = Channel number
%2 = Axis name
Definitions: The CC_Master and/or CC_Slave axes were not at standstill when the coupling was switched on.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Input G601 for path axes or enter a STOPRE before the CC_COPON command.
Program Continuation: Clear alarm with the RESET key. Restart part program

75070 Channel %1 wrong machine data for collision protection %2

Parameters: %1 = Channel number
%2 = Axis name
Definitions: Incorrect machine data for collision protection.
Reactions: - Interpreter stop
Remedy: Correct machine data. The axes must be either both rotary axes or both linear axes!
Program Continuation: Clear alarm with the RESET key. Restart part program

75071 Channel %1 collision monitoring axis %2

Parameters: %1 = Channel number
%2 = Axis name
Definitions: Collision monitor has responded.
Reactions: - Alarm display.
Remedy: Traverse the axis out of the danger area in manual mode.
Program Continuation: Clear alarm with the RESET key. Restart part program

75100 Too many analog axes configured

Definitions: More than 3 NC axes are configured as analog axes in machine data 63530 $MA_ANALOG_AXIS.
Reactions: - Alarm display.
Remedy: Reduce the number of analog axes.
Program Continuation: Switch control OFF - ON.

75110 Axis %1 reached drift limit

Parameters: %1 = Axis name
Definitions: The automatic drift compensation has reached the value set in MD 36710 $MA_DRIFT_VALUE.
Reactions: - Alarm display.
Remedy: Increase either the value in MD 36710 or the fixed drift value in MD 36720 $MA_DRIFT_VALUE.
Program Continuation: Clear alarm with the RESET key. Restart part program

75200 Channel %1 wrong MD configuration, %2 incorrect

Parameters: %1 = Channel number
%2 = Machine data
Definitions: A window in machine data %2 has been detected during power–up in the machine data of the handling transformation package.
Reactions: - Alarm display.
Remedy: Configure machine data.
Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

75210 Channel %1 number of axes/axis assignment inconsistent

Parameters: %1 = Channel number

Definitions: The number of axes specified in MD TRAFO 6_NUM_AXES and the number of axes specified in MD TRAFO_AXES_ IN_1 is inconsistent or the assignment of axis in MD TRAFO_AXES_IN_1 is incorrect. This alarm is output if the transformation is selected via TRAORI.

Reactions: - Interpreter stop
- NC Start disable in this channel.

Remedy: Configure machine data.

Program Continuation: Clear alarm with the RESET key. Restart part program

75250 Channel %1 tool parameters incorrect

Parameters: %1 = Channel number

Definitions: The tool parameters are not the same as the settings for the handling transformation package (checked in interpreter).

Reactions: - Interpreter stop
- NC Start disable in this channel.

Remedy: Configure machine data.

Program Continuation: Clear alarm with the Delete key or NC START.

75255 Channel %1 working area error

Parameters: %1 = Channel number

Definitions: The programmed point is not within the working range of the kinematic (checked in interpreter).

Reactions: - Interpreter stop

Remedy: Correct position.

Program Continuation: Clear alarm with the RESET key. Restart part program

75260 Channel %1 block %2 tool parameters incorrect

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The tool parameters are not the same as the settings for the handling transformation package (checked during preprocessing run).

Reactions: - Alarm display.

Remedy: Correct tool parameters.

Program Continuation: Clear alarm with the RESET key. Restart part program

75265 Channel %1 block %2 working area error

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed point is not within the working range of the kinematic (checked during preprocessing run).

Reactions: - Interpreter stop
- NC Start disable in this channel.

Remedy: Correct position.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

75270 Channel %1 tool parameters incorrect

Parameters: %1 = Channel number

Definitions: The tool parameters are not the same as the settings for the handling transformation package (checked in interpolation).

Reactions: - NC Stop on alarm.
- NC Start disable in this channel.

Remedy: Correct tool parameters.

Program Continuation: Clear alarm with the RESET key. Restart part program

75275 Channel %1 block %2 working area error

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed point is not within the working range of the kinematic (checked during main run).

Reactions: - Interpreter stop
- NC Start disable in this channel.

Remedy: Correct position.

Program Continuation: Clear alarm with the RESET key. Restart part program

75451 Error with definition of setpoint switchover

Definitions: Exchange axis number and machine axis number are identical, gaps have been defined, or the machine axis number is the machine axis number of an inactive machine axis of the system.

Reactions: - The NC switches to follow-up mode.

Remedy: Enter another axis number for the setpoint exchange in MD 63750.

Program Continuation: Switch control OFF - ON.

75452 Axis %1 setpoint switchover not possible in the current state

Parameters: %1 = Axis name

Definitions: The axes included in the setpoint exchange group are not all stationary (DB3x.DBB61.4)
No ModeGroupReady signal
One of the two axes in the setpoint exchange group is already configured in another active setpoint exchange. Enter a valid value in drive MD 63750: CTRLOUT_CHANGE_TAB[0.2] for the setpoint exchange to be activated equals zero. A new exchange has been requested via PLC before the function has been returned to its initial state.

Reactions: - Alarm display.

Remedy: Enter another axis number for the setpoint exchange in MD 63750.

Program Continuation: Clear alarm with the RESET key. Restart part program

75500 Channel %1 wrong configuration

Parameters: %1 = Channel number

Definitions: There are two causes of this alarm:
- No geometry axis has been defined
- The option "Software cam" is set.

Reactions: - Alarm display.

Remedy: Modify configuration.

Program Continuation: Clear alarm with the RESET key. Restart part program

© Siemens AG, 2002. All rights reserved

75600 Channel %1 retrace support: wrong MD configuration. Error code %2

Parameters: %1 = Channel number
%2 = Error code

Definitions: The following errors were detected in the machine data of the retrace support function when starting up:
Error code = 4
Machine data MM_NUM_CC_BLOCK_ELEMENTS must be increased.
Error code = 5
Insufficient compile cycle heap memory available.
Adjust machine data $MC_RESU_RING_BUFFER_SIZE, $MC_RESU_SHARE_OF_CC_HEAP_MEM and $MC_MM_NUM_CC_HEAP_MEM.
Error code = 6
The machine data $MN_ASUP_START_MASK and $MN_ASUP_START_PRIO_LEVEL are not set correctly.

Reactions: - Alarm display.
- NC Start disable in this channel.

Remedy: Correct machine data.

Program Continuation: Switch control OFF - ON.

75601 Channel %1 block %2 invalid parameter in CC_PREPRE()

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Only the values –1, 0, 1 are valid for the parameter.

Reactions: - Alarm display.
- Interpreter stop

Remedy: Modify part program.

Program Continuation: Clear alarm with the RESET key. Restart part program

75605 Channel %1 retrace support: internal error, error code %2

Parameters: %1 = Channel number
%2 = Error code

Definitions: With this alarm, RESU-internal error states are displayed which, together with the transferred error number, provide information on the error cause and error location.

Reactions: - Alarm display.
- NC Start disable in this channel.

Remedy: If this error occurs, please contact us on the SINUMERIK Hotline of the SIEMENS AG, specifying the error number.

Program Continuation: Clear alarm with the RESET key. Restart part program

75606 Channel %1 retraceable contour was shortened

Parameters: %1 = Channel number

Definitions: The block search buffer is full. Therefore the retraceable contour had to be shortened.

Reactions: - Alarm display.

Remedy: Adjust machine data $MC_RESU_RING_BUFFER_SIZE, $MC_RESU_SHARE_OF_CC_HEAP_MEM and $MC_MM_NUM_CC_HEAP_MEM.

Program Continuation: Clear alarm with the Delete key or NC START.

© Siemens AG, 2002. All rights reserved

75607 Channel %1 resynchronisation not possible

Parameters: %1 = Channel number

Definitions: The block search triggered by the compile cycle has been terminated with an error. It can have the following cause: The control is not in the correct operating mode, e.g. in JOG–AUTO instead of in AUTO.

Reactions: - Alarm display.

Remedy: Switch the control to the AUTO operating mode and restart resynchronisation.

Program Continuation: Clear alarm with the Delete key or NC START.

100300 xxx not found

Definitions: The search term entered in a list image (e.g. general machine data) was not found.

Reactions: - -

Remedy: -

Program Continuation: Internal

100301 The table cannot be completely generated

Definitions: The list image could not be generated due to insufficient memory.

Reactions: - -

Remedy: System error, a rebooting may be necessary.

Program Continuation: Internal

100302 No data available - or no access authorization

Definitions: The list image can not be generated, as this data is currently not available. Example: Local user data is not defined.

Reactions: - -

Remedy: -

Program Continuation: Internal

100303 Paging not possible

Definitions: You cannot page over, for example, axes, drives or channels, as more axes, drives or channels are not configured.

Reactions: - -

Remedy: -

Program Continuation: Internal

100350 Display MD saved

Definitions: - The display machine data is saved via the soft key "Save" in the operating area start-up, image machine data - display machine data.

- The display machine data is saved in the start-up basic display after pressing the soft key "LCD brighter" or "LCD darker" (this setting will remain at the next start-up).

- As of SW 4.1: If the display options are changed in the machine data images, the change will be saved in the display machine data that is not visible to the user.

Reactions: - -

Remedy: -

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

100351 Display MD take-over not possible

Definitions: Saving the display machine data was rejected by the NCK.
Reactions: - -
Remedy: -
Program Continuation: Switch control OFF - ON.

101000 No connection to the PLC!

Definitions: The connection to the PLC cannot be made while booting, e.g. wrong PLC basic program.
Reactions: - -
Remedy: -
Program Continuation: Internal

101001 The PLC system status list cannot be read!

Definitions: After the connection has been made, the system status list cannot be read.
Reactions: - -
Remedy: Switch controller off/on
Program Continuation: Internal

101002 Invalid password!

Definitions: The password entered is wrong.
Reactions: - -
Remedy: Enter a valid password.
Program Continuation: Internal

101003 Password for %1 is set!

Parameters: %1 = Access-level system, manufacturer, service or user.
Definitions: The password for system, manufacturer, service or user was set successfully.
Reactions: - -
Remedy: -
Program Continuation: Internal

101004 Password for %1 is set!

Parameters: %1 = Access-level system, manufacturer, service or user.
Definitions: The password for system, manufacturer, service or user was changed successfully.
Reactions: - -
Remedy: -
Program Continuation: Internal

101005 The passwords do not match!

Definitions: When the password was changed, the password entered first does not match the one entered second.
Reactions: - -
Remedy: Enter a valid password.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

101006 Password is deleted!

Definitions: The password was deleted via the soft key "Delete password".
Reactions: - -
Remedy: Enter password.
Program Continuation: Internal

101007 Password is not set!

Definitions: To delete the password, a higher access authorization is required (at least user).
Reactions: - -
Remedy: Set the password with a higher access level.
Program Continuation: Internal

101008 Current access level: %1

Parameters: %1 = Access-level system, manufacturer, service or user.
Definitions: When selecting the alarm image, the current access level is displayed: system, manufacturer, service or user or keyswitch positions 3/2/1/0.
Reactions: - -
Remedy: -
Program Continuation: Internal

101013 Input error - see help - (i)-key

Definitions: PLC status A syntax error occurred while entering a value in the PLC status.
The input syntax is explained in a help image.
Reactions: - -
Remedy: -
Program Continuation: Internal

101016 Error: The operand address is greater than 65535!

Definitions: The value range of the operand address was exceeded.
Reactions: - -
Remedy: Use a smaller value range for the operand address.
Program Continuation: Internal

101017 No PLC input screen forms found!

Definitions: There are no *.plc input screen forms in the target system.
Reactions: - -
Remedy: -
Program Continuation: Internal

101018 Reading-in only possible in an active PLC status!

Definitions: The current PLC status mode is not active, e.g. if the soft key "Change" was pressed.
Reactions: - -
Remedy: Switch the PLC status to active.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

101100 No access authorization!

Definitions: The access level set is too low to open the selected window.
Reactions: - -
Remedy: Enter a higher password.
Program Continuation: Internal

101111 No axes configured!

Definitions: Due to an incomplete start-up, the image "Service axis" or "Axis machine data" cannot be selected.
Reactions: - -
Remedy: Complete start-up.
Program Continuation: Internal

101112 No drives configured!

Definitions: Due to an incomplete start-up, the image "Service drive" cannot be selected.
Reactions: - -
Remedy: Complete start-up.
Program Continuation: Internal

101113 No channels configured!

Definitions: Due to an incomplete start-up, the image "Channel machine data" cannot be selected.
Reactions: - -
Remedy: Complete start-up.
Program Continuation: Internal

101114 No MSD configured!

Definitions: Due to an incomplete start-up or missing MSD drives, the image "MSD machine data" cannot be selected.
Reactions: - -
Remedy: -
Program Continuation: Internal

101115 No FDD configured!

Definitions: Due to an incomplete start-up or missing FDD/SLM drives, the image "FDD machine data" cannot be selected.
Reactions: - -
Remedy: -
Program Continuation: Internal

101130 Error return value not defined: 00h 00h

Definitions: A function was called up in the start-up area that then, for unknown reasons, could not be executed.
Reactions: - -
Remedy: When provided with the issued digits, the service may be able to help.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

101131 No servo disable at PI start

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101132 Impermissible execution argument value

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101133 MDx120 CURRCTRL_GAIN could not be calculated

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101134 MDx407 SPEEDCTRL_GAIN_1 could not be calculated

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101135 MDx409 SPEEDCTRL_INTEGRATOR_TIME_1 could not be calculated

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101136 MDx150 FIELDCTRL_GAIN could not be calculated

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101137 MDx141 MAGNETIZING_REACTANCE=0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101138 MDx139/MDx140 MD_STATOR-/ROTOR_LEAKAGE_REACTANCE=0

Definitions: -
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

101139 MDx134 MOTOR_NOMINAL_FREQUENCY=0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101140 MDx138 ROTOR_COLD_RESISTANCE = 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101141 MDx117 MOTOR_INERTIA = 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101142 MDx146 < MDx142 MOTOR_MAX_ALLOWED_SPEED < FIELD_WEAKENING_SPEED

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101143 MDx142 FIELD_WEAKENING_SPEED = 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101144 MDx118 MOTOR_STANDSTILL_CURRENT = 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101145 MD1104/1118 MOTOR_MAX_CURRENT/MOTOR_STANDSTILL_CURRENT > 900.0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

101146 Boot file(s) saved

Definitions: Saving the boot file in the start-up area, image drive machine data, was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101147 Boot file(s) deleted

Definitions: Deleting the boot file in the start-up area, image drive machine data, was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101148 Controller MD calculated

Definitions: Calculating the controller data in the start-up area, image drive machine data, was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101150 MD set active

Definitions: Activation of the machine data in the start-up area, machine data images, was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101151 Start-up successful

Definitions: In the start-up area, image NC start-up, one of the three functions - Normal booting - Booting with default values - Start of the software update was successfully initiated.
Reactions: - -
Remedy: -
Program Continuation: Internal

101153 MMC-NCK communication faulty %1 %2

Parameters: %1 = Error class
%2 = Error code
Definitions: In the start-up area the soft key "Calculate controller data", for example, was pressed.
An unspecific error message is sent from the NCK or drive as acknowledgement for this function call.
By using the two hexadecimal values (error class, error code), the start-up engineer can perform an error diagnosis.
Reactions: - -
Remedy: -
Program Continuation: Internal

101154 PI service was rejected

Definitions: The current status of the NKC/drive does not permit the function that was selected.
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: See the Installation and Start-up Guide.
Program Continuation: Internal

101155 Path %1 not available

Parameters: %1 = Path
Definitions: During a file function, e.g.saving boot files, an attempt was made to access a non-existent path.
Reactions: - -
Remedy: Switch the control OFF/ON or see the Installation and Start-up Guide.
Program Continuation: Switch control OFF - ON.

101156 Impermissible function

Definitions: The selected function is impermissible.
Reactions: - -
Remedy: See the Installation and Start-up Guide.
Program Continuation: Internal

101157 File %1 not available

Parameters: %1 = File name
Definitions: In the start-up area the soft key "delete boot files", for example, was pressed, although no boot files are yet available.
Reactions: - -
Remedy: -
Program Continuation: Internal

101158 Function not permitted in the current operating state.

Definitions: The drive's current status does not permit this function.
Reactions: - -
Remedy: -
Program Continuation: Internal

101159 Remote block in wrong state

Definitions: The drive's current status does not permit this function.
Reactions: - -
Remedy: -
Program Continuation: Internal

101160 Date and time of the PLC set

Definitions: In the PLC status, the time or date was changed.
Reactions: - -
Remedy: -
Program Continuation: Internal

101161 The drive is not in cyclic mode!

Definitions: The start-up is not completely carried out, therefore, the function "Calculate motor data" cannot be selected.

© Siemens AG, 2002. All rights reserved

Reactions: - -
Remedy: -
Program Continuation: Internal

101162 MDx134/MDx400 MOTOR_NOMINAL_FREQUENCY/MOTOR_RATED_SPEED impermissible

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101163 MDx130 MOTOR_NOMINAL_POWER <= 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101164 MDx132 MOTOR_NOMINAL_VOLTAGE <= 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101165 MDx103 MOTOR_NOMINAL_CURRENT <= 0

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101166 MDx129 POWER_FACTOR_COS_PHI impermissible

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101167 MDx134/MDx400 MOTOR_NOMINAL_FREQUENCY/MOTOR_RATED_SPEED impermissible

Definitions: -
Reactions: - -
Remedy: -
Program Continuation: Internal

101168 Warning MDx142 FIELD_WEAKENING_SPEED < MDx400 MOTOR_RATED_SPEED

Definitions: -
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

101200 Safety-Integrated data copied

Definitions: The SI data copying function was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101201 Safety-Integrated data confirmed

Definitions: The SI data confirmation function was successful.
Reactions: - -
Remedy: -
Program Continuation: Internal

101202 SI data is copied from axis %1 to drive %2

Parameters: %1 = Axis name
%2 = Drive number
Definitions: This message is output during the SI data copying function.
Reactions: - -
Remedy: -
Program Continuation: Internal

101203 SI data not completely copied

Definitions: An error occurred during the SI data copying function; this caused the SI data to be copied incompletely or not at all.
Reactions: - -
Remedy: -
Program Continuation: Internal

101204 SI data not confirmed

Definitions: The SI data confirmation function was not executed because an error occurred during the processing.
Reactions: - -
Remedy: -
Program Continuation: Internal

101205 Drive data changed? -> Don't forget to save the boot files!

Definitions: When exiting the drive machine data images, the operator is reminded to save the boot files so that drive machine data that may have been changed is not lost.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

101206 Search action running, pleas wait

Definitions: The search function was initiated in the machine data images.
Reactions: - -
Remedy: -
Program Continuation: Internal

101207 Position at $MN_

Definitions: A list image, e.g. general machine data, was selected.
The MMC100 attempts to position at the datum selected last in this image.
Reactions: - -
Remedy: -
Program Continuation: Internal

101208 SI data confirmed: Axis 1

Definitions: In the start-up area the function "Confirm Safety Integrated Data" has been started.
The message was output during this function in order to provided the user with an acknowledgement regarding the operation of the function.
Reactions: - -
Remedy: -
Program Continuation: Internal

101209 SI data confirmed: Drive 1

Definitions: In the start-up area the function "Confirm Safety Integrated Data" has been started.
The message was output during this function in order to provided the user with an acknowledgement regarding the operation of the function.
Reactions: - -
Remedy: -
Program Continuation: Internal

101210 The machine data is being prepared for display

Definitions: A list image in the machine data was selected for which display options are active.
This image's machine data is individually checked to see whether it is permitted to be displayed.
Reactions: - -
Remedy: -
Program Continuation: Internal

101211 NCK address not changed!

Definitions: An attempt was made to change the bus address of the NCK in the start-up area.
The change was rejected by the NCK; the reason is unknown.
Reactions: - -
Remedy: -
Program Continuation: Internal

101212 NCK address changed!

Definitions: The NCK's bus address was set to the specified value.
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

101213 Invalid NCK address.

Definitions: The value specified for the new NCK bus address is too large.
Reactions: - -
Remedy: -
Program Continuation: Internal

101214 Initialization of this window failed!

Definitions: An attempt was made to select the image of the NCK address in the start-up area.
An error occured while determining the nodes on the bus.
Due to inconsistent data, this image cannot be displayed.
Reactions: - -
Remedy: -
Program Continuation: Internal

101300 Please wait - language is being changed

Definitions: In the start-up area the soft key "Change language" was pressed.
The screen content is being restructured.
Reactions: - -
Remedy: -
Program Continuation: Internal

103000 There is no correction block on the NCK

Definitions: The following is valid up to SW 3.x: The correction block window cannot be opened. No program correction is possible in the operating mode "Machine".
The following is valid up to SW 4.1: The correction block window can be opened in the "Stop program" state.
The edit program that is currently being executed is opened.
The program part that has not yet been recorded by the NCK's program processing can be changed permanently.
Reactions: - -
Remedy: -
Program Continuation: Internal

103001 Selection is only possible after being enabled or in the RESET state.

Definitions: In order to execute the desired function, the current channel has to be in the RESET state, e.g. "Program selection".
Reactions: - -
Remedy: Trigger reset.
Program Continuation: Internal

103002 Copying to the clipboard is not possible

Definitions: The NCK rejects a copy-action to the clipboard because, for example, no more memory is available or the maximum manageable amount of programs has been exceeded.
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: Delete the programs not currently required.
Program Continuation: Internal

103003 MDI clipboard cannot be deleted

Definitions: The NCK rejects a deletion of the clipboard because the clipboard is currently being executed.
Reactions: - -
Remedy: Wait until the MDI program has been executed or trigger an NC reset.
Program Continuation: Internal

103004 Block search not possible

Definitions: The search run cannot be started because the channel is active.
Reactions: - -
Remedy: Wait until the program has been executed or trigger an NC reset.
Program Continuation: Internal

103006 Block search without calculation only possible on the main program level.

Definitions: No subroutine calls can be processed during a block search without calculation.
Reactions: - -
Remedy: If subroutines are to be processed, a search run with calculation has to be performed.
Program Continuation: Internal

103007 Ending overstore is not possible in this channel state.

Definitions: Overstore cannot be ended because the channel is still active.
Reactions: - -
Remedy: Wait until the overstore procedure has finished or trigger an NC reset.
Program Continuation: Internal

103008 Block search in channel %1 started - please wait

Parameters: %1 = Channel number
Definitions: The started block search is not yet finished.
Reactions: - -
Remedy: Text is deleted automatically after the end of the block search.
Wait until the block search has finished or trigger an NC reset.
Program Continuation: Internal

103009 Conflict between search type and search target!

Definitions:
- The search target entered is not compatible with the search type:
- The block number initial character "N" or ":" is missing,
- only the digits 0 to 9 are allowed.

Reactions: - -
Remedy: Correct the entry to correspond with the type.
- The following is valid up to SW 3.x: The faulty entry is deleted, and the cursor proceeds to the next field.
- As of SW 4.1: The faulty entry is retained, and the cursor remains at the field.

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

103010 Invalid file name

Definitions: The file name entered:
- Must not have more than 32 characters (letters, digits, underscore; including the block and program ID:_N_ and _MPF), amounts to 25 assignable characters.
- Cannot have any separators (e.g. /).
- Must have letters at the first and second position.

Reactions: - -
Remedy: -
Program Continuation: Internal

103011 No program selected - block search finished

Definitions: At the moment, no program is selected, therefore, no search run is possible.
Reactions: - -
Remedy: Select a program.
Program Continuation: Internal

103014 Please reference the axis first

Definitions: The reference point approach was not yet performed or finished.
Reactions: - -
Remedy: Perform reference point approach.
Program Continuation: Internal

104000 Current tool not found

Definitions: If the cursor in the window "Magazine list" is at a blank line, then no tool will be found after pressing the soft key "Tool data".
Reactions: - -
Remedy: Place the cursor on the tool.
Program Continuation: Internal

104001 No other tools available

Definitions: After pressing the soft key "T No. +" or "T No. -" the next-highest or next-lowest tool number is searched for.
If no other tools are available, this message will be output.
Reactions: - -
Remedy: -
Program Continuation: Internal

104002 No other cutting edges available

Definitions: After pressing the soft key "D No. +" or "D No. -" the next-highest or next-lowest cutting edge is searched for.
If no other cutting edges are available, this message will be output.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

104003 There aren't any tools at all

Definitions: No tools were set up.
Reactions: - -
Remedy: Set up tools.
Program Continuation: Internal

104004 No tool selected

Definitions: After pressing the soft key "Current T+D No." in the "Tool overview" window, no tool was found because no subroutine is active or in the active subroutine no tool is selected.
Reactions: - -
Remedy: Select a tool.
Program Continuation: Internal

104005 No cutting edge selected

Definitions: A tool has been selected, but not a cutting edge.
Reactions: - -
Remedy: Select a cutting edge.
Program Continuation: Internal

104006 No TO area available in the current channel

Definitions: No TO area is assigned to the current channel.
Reactions: - -
Remedy: Assign the current channel a TO area via the machine data, or switch channels.
Program Continuation: Internal

104007 Error during tool creation

Definitions: The tool could not be created, as, for example, the maximum number of possible tools has already been reached.
Reactions: - -
Remedy: Extend the maximum number of tools via the machine data, or delete unnecessary tools.
Program Continuation: Internal

104008 Error during cutting edge creation

Definitions: No new cutting edge could be created, as, for example, the maximum number of cutting edges has already been reached.
Reactions: - -
Remedy: Delete unnecessary cutting edges.
Program Continuation: Internal

104011 Error during tool deletion

Definitions: The tool cannot be deleted. It may currently be active.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

104012 No master spindle available

Definitions: No master spindle was configured.
Reactions: - -
Remedy: Configure a master spindle via the machine data.
Program Continuation: Internal

104014 Incorrect entry

Definitions: The entered value is impermissible, e.g., it lies outside the input limits.
Reactions: - -
Remedy: Please observe the value range.
Program Continuation: Internal

104015 Number of parameters per cutting edge is zero

Definitions: The number of parameters per cutting edge was not configured.
Reactions: - -
Remedy: Configure the number of parameters per cutting edge via the machine data.
Program Continuation: Internal

104016 No spindle available

Definitions: No spindle was configured.
Reactions: - -
Remedy: Configure a spindle via the machine data.
Program Continuation: Internal

104018 Tool not available

Definitions: The specified tool does not exist.
Reactions: - -
Remedy: -
Program Continuation: Internal

104019 Tool type not available

Definitions: The specified tool type does not exist.
Reactions: - -
Remedy: Specify a valid tool type.
Program Continuation: Internal

104020 No empty location found

Definitions: There is no empty location with the specified search parameters.
Reactions: - -
Remedy: Correct search parameters.
Program Continuation: Internal

104023 Error during tool loading

Definitions: An error occurred while a tool was being loaded; the procedure has been aborted.
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: Check loader.
Program Continuation: Internal

104024 Error during tool unloading

Definitions: An error occurred while a tool was being unloaded; the procedure has been aborted.
Reactions: - -
Remedy: Check unloader.
Program Continuation: Internal

104025 No magazine configured

Definitions: No magazine was configured.
Reactions: - -
Remedy: Configure a magazine via the machine data.
Program Continuation: Internal

104026 Notice: Tool is being loaded!

Definitions: Note regarding loading process.
Reactions: - -
Remedy: Wait until the loading procedure is terminated.
Program Continuation: Internal

104027 Notice: Tool is being unloaded!

Definitions: Note regarding unloading process.
Reactions: - -
Remedy: Wait until the unloading procedure is terminated.
Program Continuation: Internal

104029 Loading to this location not possible

Definitions: The location may already be occupied.
Reactions: - -
Remedy: Select another loading location.
Program Continuation: Internal

104030 No more data available

Definitions: All existing data was already displayed.
Reactions: - -
Remedy: -
Program Continuation: Internal

105000 Error x y

Definitions: System-internal error.
A memory-access has failed - should not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

105001 Cycle description sc.com not available

Definitions: The cycle description sc.com was not found in the NCK file system.
Reactions: - -
Remedy: -
-
Program Continuation: Internal

105002 File xxx already exists

Definitions: The file name entered is already in this directory.
Reactions: - -
Remedy: -
Program Continuation: Internal

105003 Workpiece xxx already exists

Definitions: The workpiece name entered is already in this directory.
Reactions: - -
Remedy: -
Program Continuation: Internal

105004 Clipboard empty! To PASTE, COPY first

Definitions: No file could be inserted, as no file was copied to the clipboard.
Reactions: - -
Remedy: -
Program Continuation: Internal

105005 Only workpieces can be inserted here

Definitions: The file type of the previously copied file is not _wpd and cannot be inserted into the workpiece directory.
Reactions: - -
Remedy: -
Program Continuation: Internal

105006 Only files can be inserted here

Definitions: A file of the file type _wpd, i.e. a workpiece, was copied from the workpiece directory and an attempt was made to insert it into a program directory.
Reactions: - -
Remedy: -
Program Continuation: Internal

105007 No name specified

Definitions: -
Reactions: - -
Remedy: Wait until the loading procedure is terminated.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

105008 Memory error while writing a cycle call

Definitions: The resources of the MMC100 are exhausted.
No more dynamic memory available.
Reactions: - -
Remedy: -
Program Continuation: Internal

105009 No write-authorization for the data

Definitions: The file is write-protected.
Reactions: - -
Remedy: -
Program Continuation: Internal

105010 No data selected

Definitions: The cursor is not placed on a valid file name.
Reactions: - -
Remedy: -
Program Continuation: Internal

105011 Cycle description cov.com not available

Definitions: The cycle description sc.com was not found in the NCK file system.
Reactions: - -
Remedy: -
Program Continuation: Internal

105012 The program is not or only partially editable (NC reset)

Definitions: The selected program is currently being executed.
Reactions: - -
Remedy: -
Program Continuation: Internal

105013 The copied data can be inserted with the soft key "Paste"

Definitions: The copied data is in the clipboard and can be inserted anywhere via the soft key "Paste".
Reactions: - -
Remedy: -
Program Continuation: Internal

105014 Error while copying!

Definitions: The file could not be copied.
Reactions: - -
Remedy: -
Program Continuation: Internal

105015 Error while renaming!

Definitions: The file could not be renamed.
Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

105016 Error while deleting!

Definitions: The file could not be deleted.
Reactions: - -
Remedy: -
Program Continuation: Internal

105017 Selection is only possible after being enabled or in the RESET state.

Definitions: The selected program is either being currently executed or was not yet enabled.
Reactions: - -
Remedy: -
Program Continuation: Internal

105018 Error during program generation!

Definitions: The program could not be created; there may no longer be sufficient memory.
Reactions: - -
Remedy: -
Program Continuation: Internal

105019 Error while opening a window!

Definitions: The window could not be opened.
System error that should not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105020 Error while closing a window!

Definitions: The window could not be closed.
System error that should not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105021 Error during workpiece generation!

Definitions: The workpiece could not be created.
Reactions: - -
Remedy: -
Program Continuation: Internal

105022 Error during enabling!

Definitions: The program could not be enabled.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

105023 File exists already!

Definitions: The file cannot be copied to this directory, as a file with the same name is already here.
Reactions: - -
Remedy: -
Program Continuation: Internal

105024 Check values! - At least 1 value lies outside the input limits!

Definitions: An invalid value was entered in the cycle parameter image.
Reactions: - -
Remedy: -
Program Continuation: Internal

105025 Please wait, the directory is being prepared!

Definitions: The data required for the display of the directory is being determined.
Reactions: - -
Remedy: -
Program Continuation: Internal

105026 Notice! The simulated program and edited program are not the same!

Definitions: The program being executed is not identical to the program opened in the editor.
Reactions: - -
Remedy: -
Program Continuation: Internal

105030 Please wait, renumbering is being carried out!

Definitions: The part program's blocks are serially numbered.
Reactions: - -
Remedy: -
Program Continuation: Internal

105031 Renumbering was aborted!

Definitions: Renumbering of the part program was aborted.
Insufficient part program memory may be the cause of the error.
Reactions: - -
Remedy: -
Program Continuation: Internal

105032 Renumbering finished!

Definitions: Renumbering of the part program was completed without errors.
Reactions: - -
Remedy: -
Program Continuation: Internal

105041 Block number is too large!

Definitions: The set increment and the size of the program cause the block number to be greater than 999999.

© Siemens AG, 2002. All rights reserved

Reactions: - -
Remedy: -
Program Continuation: Internal

105042 Impermissible block number!

Definitions: The first block number is less than 0 or greater than 999999.
Reactions: - -
Remedy: -
Program Continuation: Internal

105043 Impermissible increment!

Definitions: The increment was entered as a negative.
Reactions: - -
Remedy: -
Program Continuation: Internal

105050 Please wait: Graphic is being output!

Definitions: The help displays are being prepared for display.
Reactions: - -
Remedy: -
Program Continuation: Internal

105051 Output of dynamic long texts for cycle parameterization

Definitions: The dynamic long texts for the cycle parameterization are output here.
Reactions: - -
Remedy: -
Program Continuation: Internal

105052 Error in the cycle description of the <xxx>!

Definitions: The cycle description sc.com, uc.com contains a line that cannot be interpreted.
This line is output via <xxx>.
Reactions: - -
Remedy: -
Program Continuation: Internal

105053 No cycle available in current line!

Definitions: The editor's cursor is in a line that does not contain a cycle.
A recompilation is not possible.
Reactions: - -
Remedy: -
Program Continuation: Internal

105054 Error during cycle description call!

Definitions: One of the sc.com, cov.com cycle description files contains a non-interpretable parameter.
IInitilization of the cycles is aborted.

© Siemens AG, 2002. All rights reserved

Reactions: - -
Remedy: -
Program Continuation: Internal

105060 Please wait: Initilization of the cycle support

Definitions: The cycle description files are interpreted and prepared for display on the screen.
Reactions: - -
Remedy: -
Program Continuation: Internal

105061 Error when opening the file!

Definitions: The specified file could not be opened.
System-internal error that does not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105062 Error when closing the file!

Definitions: The specified file could not be closed.
System-internal error that does not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105063 Error when positioning in the file %1!

Parameters: %1 = File name
Definitions: No positioning could be done in the specified file.
System-internal error that does not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105064 Error when reading the file!

Definitions: The specified file could not be read.
System-internal error that does not occur in normal operation.
Reactions: - -
Remedy: -
Program Continuation: Internal

105070 Please wait: Initialization of the simulation started!

Definitions: The graphic travel path is being initialized.
Reactions: - -
Remedy: -
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

105075 Not enough axes in the current channel? > Contour with default axis names: X, Z !

Definitions: The default axis names for the required axes are used.
Reactions: - -
Remedy: -
Program Continuation: Internal

109001 No switchover: Switchover disable set in current PLC

Definitions: MMC would like to go offline from this NCU.
The MMC switchover is disabled in the MMC PLC online interface in DB19. (MMCx_SHIFT_LOCK = TRUE, x = 1,2)
Reactions: - -
Remedy: -
Program Continuation: Internal

109002 No switchover: Target PLC occupied, try again

Definitions: MMC would like to go online to this NCU.
MMC has called the target PLC and is awaiting acknowledgement.
No acknowlegdement for MMC since MMC parameter interface in DB19 is assigned by another MMC.
Reactions: - -
Remedy: Repeat the operation at a later point in time, as the MMC parameter interface in DB19 is only temporarily occupied.
Program Continuation: Internal

109003 No switchover: Switchover disable set in target PLC

Definitions: MMC would like to go online to this NCU.
The MMC switchover is disabled in the MMC PLC online interface in DB19. (MMCx_SHIFT_LOCK = TRUE, x = 1,2)
Reactions: - -
Remedy: The MMC switchover is disabled/enabled in the machine manufacturer's PLC program. Reference to the machine manufacturer's documentation.
Program Continuation: Internal

109004 No switchover: PLC occupied by higher-priority MMC's

Definitions: The MMC is attempting to switch to an NCU that is occupied by two higher-priority MMC's.
Reactions: - -
Remedy: Switch one of the two higher-priority MMC's to another NCU.
Program Continuation: Internal

109005 No switchover: No displaceable MMC at the target PLC

Definitions: MMC would like to go online to this NCU.
At this NCU, two MMC's are online on which uninterruptable processes are active (e.g.: data transfer between MMC and NCU).
Reactions: - -
Remedy: Wait until at least one of the two MMC's can be displaced and repeat the switchover.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

109006 No switchover: Selected channel invalid

Definitions: At this NCU, the MMC was switched to a channel that does not exist there.
Reactions: - -
Remedy: Set up the channel or adapt the parameterization of the NETNAMES.INI.
Program Continuation: Internal

109007 Channel switchover running

Definitions: The channel switchover has been initiated.
A different MMC may have to be displaced.
Reactions: - -
Remedy: -
Program Continuation: Internal

109008 Activation is running

Definitions: The switchover from the passive operating mode to the active operating mode has been initiated.
Reactions: - -
Remedy: -
Program Continuation: Internal

111001 Non-interpretable step in line %1

Parameters: %1 = Line number
Definitions: The step is not an element of ManualTurn.
Easystep sequencer is not loaded.
Reactions: - Alarm display.
Remedy: Delete the program step or change the program in the operating area PROGRAMS in the SINUMERIK 840D or 810D (CNC mode).
Program Continuation: Internal

111002 Insufficient memory, abort in line %1

Parameters: %1 = Line number
Definitions: Easystep sequencer has too many steps.
Easystep sequencer is not loaded.
Reactions: - Alarm display.
Remedy: Change the program in the operating area PROGRAMS in the SINUMERIK 840D or 810D (CNC mode).
Program Continuation: Internal

111003 ManualTurn: %1

Parameters: %1 = Error code
Definitions: Internal system message over the ManualTurn operator panel.
Reactions: - Alarm display.
Remedy: Acknowledge error and inform Siemens.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

111004 File faulty or not available: %1

Parameters: %1 = File/Contour name
Definitions: Easystep sequencer cannot interpret a step with contour programming.
Contour not in the directory.
Reactions: - Alarm display.
Remedy: Load contour into the directory.
Program Continuation: Internal

111005 Error when interpreting the contour %1

Parameters: %1 = Contour name
Definitions: Contour is faulty.
Reactions: - Alarm display.
Remedy: Check the contour's machining sequence.
Program Continuation: Internal

111006 Maximum number of contour elements exceeded %1

Parameters: %1 = Contour name
Definitions: The maximum permissible number of 50 contour elements was exceeded when interpreting the machining sequence of a contour.
Reactions: - Alarm display.
Remedy: Check the contour's machining sequence and, if necessary, edit it.
Program Continuation: Internal

111007 Error in line %1 %2

Parameters: %1 = Line number
%2 = Error description
Definitions: -
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Eliminate the corresponding error.
Program Continuation: Internal

111008 Spindle not synchronized

Definitions: Spindle not synchronized.
Reactions: - Alarm display.
Remedy: Let the spindle run at least one revolution (M3, M4).
Program Continuation: Internal

111009 Load new tool: T%1

Parameters: %1 = Tool number
Definitions: Tool change program requests a new tool.
Reactions: - Alarm display.
- NC Stop on alarm.
Remedy: Load new tool.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

111010 Teach-in interruption: Log overflow

Definitions: The Teach-in procedure was interrupted. Teach-in file is closed.

Reactions: - Alarm display.

Remedy: In the machine data 9606: $MM_CTM_SIMULATION_TIME_NEW_POS the value of the update rate is to be increased by 100 to 200 ms.

Program Continuation: Internal

111100 Wrong position programmed for the spindle

Definitions: A position beyond the range of 0 -359.999 has been programmed for a modulo axis.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Program a position in the 0 - 359.999 range.

Program Continuation: Clear alarm with the RESET key. Restart part program

111105 No measuring system available

Definitions: SPCON, SPOS or SPOSA has been programmed.

These functions require at least one measuring system. According to MD: NUM_ENCS the machine axis/spindle has no measuring system.

Reactions:
- Interpreter stop
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Retrofit a measuring system.

Program Continuation: Clear alarm with the RESET key. Restart part program

111106 No spindle stop for a block change

Definitions: The displayed spindle has been programmed as spindle or as axis even though a positioning operation is still running from the previous block (with SPOSA ... spindle positioning beyond block limits).

Example: N100 SPOSA [2] = 100 etc., N126 S2 = 1000 M2 = 04Error, if the spindle S2is still running from block N100!

Reactions:
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Before programming the spindle/axis again using the SPOSA instruction, a WAITS command should be activated in order to wait for the programmed spindle position.

Example: N100 SPOSA [2] = 100 etc., N125 WAITS (2) N126 S2 = 1000 M2 = 04

Program Continuation: Clear alarm with the RESET key. Restart part program

111107 Reference mark not found

Definitions: When referencing, the spindle turned through a greater distance than given in the axis-specific machine data 34,060 REFP_MAX_MARKER_DIST, without receiving a reference mark signal. The check is performed for spindle positioning with SPOS or SPOSA when the spindle has not previously run with speed control (S=...).

© Siemens AG, 2002. All rights reserved

Reactions: - NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Check and correct the MD 34 060 REFP_MAX_MARKER_DIST. The value entered states the distance in [mm] or [degrees] between 2 zero markers.

Program Continuation: Clear alarm with the RESET key. Restart part program

111108 No transition from speed control mode to position control mode

Definitions:
- An oriented spindle stop (SPOS/SPOSA) has been programmed or the position control of the spindle was switched on with SPCON but no spindle encoder has been defined.
- When switching on the position control, the spindle speed is greater than the limiting speed of the measuring system.

Reactions: - NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Spindle without attached encoder: Any NC language elements requiring the encoder signals must not be used.

Spindle with attached encoder: Enter the number of spindle encoders used in the MD NUM_ENCS.

Program Continuation: Clear alarm with the RESET key. Restart part program

111109 Configured positioning velocity is too high

Definitions: When referencing, the spindle turned through a greater distance than given in the axis-specific machine data 34,060 REFP_MAX_MARKER_DIST, without receiving a reference mark signal.

The check is performed for spindle positioning with SPOS or SPOSA when the spindle has not previously run with speed control (S=...).

Reactions: - NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.

Remedy: Check and correct the MD 34 060 REFP_MAX_MARKER_DIST.

The value entered states the distance in [mm] or [degrees] between 2 zero markers.

Program Continuation: Clear alarm with the RESET key. Restart part program

111110 Velocity/Speed is negative

Definitions: The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.

Reactions: - Alarm display.

Remedy: Inform the service department. Please contact the responsible Siemens regional office.

Program Continuation: Internal

111111 Setpoint speed is zero

Definitions: The programmed spindle speed setpoint is zero.

Reactions: - Alarm display.

Remedy: Set permissible spindle speed setpoint.

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

111112 Invalid gear stage

Definitions: An invalid gear stage was requested by the PLC.
Reactions: - Alarm display.
Remedy: Check the PLC program and axis-specific NC machine data.
Program Continuation: Internal

111115 Programmed position was not reached

Definitions: The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.
Reactions: - Alarm display.
Remedy: Inform the service department. Please contact the responsible Siemens regional office.
Program Continuation: Internal

111126 Absolute value minus not possible

Definitions: The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.
Reactions: - Alarm display.
Remedy: Inform the service department. Please contact the responsible Siemens regional office.
Program Continuation: Internal

111127 Absolute value plus not possible

Definitions: The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.
The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.
Reactions: - Alarm display.
Remedy: Inform the service department. Please contact the responsible Siemens regional office.
Inform the service department. Please contact the responsible Siemens regional office.
Program Continuation: Internal

111200 Spindle positioning error

Definitions: The alarms 111110, 111115, 111126, 111127 and 111200 can occur at spindle start/stop.
Reactions: - Alarm display.
Remedy: Inform the service department. Please contact the responsible Siemens regional office.
Program Continuation: Internal

111300 NC start key defective

Definitions: Acknowledgement to the PLC user program, that the NC start key is defective, i.e., NC and NO signal = 1.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Replace the key.
Program Continuation: Internal

111301 NC stop key defective

Definitions: Acknowledgement to the PLC user program, that the NC stop key is defective, i.e., NC and NO signal = 1.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Replace the key.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

111302 Spindle start key defective

Definitions: Acknowledgement to the PLC user program, that the spindle start key is defective, i.e., NC and NO signal = 1.

Reactions: - NC Start disable in this channel.
- Alarm display.

Remedy: Replace the key.

Program Continuation: Internal

111303 Spindle stop key defective

Definitions: Acknowledgement to the PLC user program, that the spindle stop key is defective, i.e., NC and NO signal = 1.

Reactions: - NC Start disable in this channel.
- Alarm display.

Remedy: Replace the key.

Program Continuation: Internal

111304 Connection to the PLC broken off

Definitions: Acknowledgement to the PLC user program, that the connection with MANUALTURN has been broken off.

Reactions: - Alarm display.

Remedy: Check the PLC user program.

Program Continuation: Internal

111305 Asynchronous subroutine was not executed

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 111306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

111306 Error when selecting or deselecting constant cutting speed

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 111306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

111307 Error when deleting handwheel offset

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 111306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

111308 Error when setting upper spindle speed limit

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 111306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

111309 Error when selecting tool

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 111306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

111310 Error when selecting zero offset

Definitions: In the asynchronous subroutine, internal settings in the NC were triggered by the operator panel. If one of the alarms from 11306 to 111310 occurs, these settings cannot be carried out.

Reactions: - Alarm display.

Remedy: Perform an NCK reset

Program Continuation: Internal

111311 NC start not possible: Deselect single block

Definitions: A program was activated with block search, while at the same time single block was active.

Reactions: - NC Start disable in this channel.
- Interface signals are set.
- Alarm display.

Remedy: Deselect single block.

Program Continuation: Internal

111400 Unknown PLC error

Definitions: The PLC has announced an error that is unknown in the operator panel.

Reactions: - NC Start disable in this channel.
- Alarm display.

Remedy: Inform Siemens.

Program Continuation: Switch control OFF - ON.

111410 Tool %1 was created

Parameters: %1 = Tool number

Definitions: When ManualTurn is booted, a check is run to see whether all standard tools are available. If this is not the case, the missing tools will be created automatically. If several tools are created, they will be output in an aggregate signal.

Meaning: Example: %1Number of the tool that was created, 5 %1First and last tool that were created, 5...16.

Reactions: - -

© Siemens AG, 2002. All rights reserved

Remedy: -
Program Continuation: Internal

111411 %1 Tool(s) can (could) not be created

Definitions: When ManualTurn is booted, a check is run to see whether all standard tools are available. If this is not the case, the missing tools will be created automatically. Hereby, the specified number of tools could not be created.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Increase machine data 18082 $MM_NUM_TOOL by the specified amount.
Program Continuation: Internal

111420 Error during the inch/metric conversion! Check all data!

Definitions: The switchover of the data for the inch/metric conversion was not completed. This alarm can only appear in the event of hardware defects.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: The following data must be checked:
Display MD's:
- MD9004: $MM_DISPLAY_RESOLUTION
- MD9600: $MM_CTM_SIMULATION_DEF_X
- MD9601: $MM_CTM_SIMULATION_DEF_Y
- MD9602: $MM_CTM_SIMULATION_DEF_VIS_AREA
- MD9603: $MM_CTM_SIMULATION_MAX_X
- MD9604: $MM_CTM_SIMULATION_MAX_Z
- MD9605: $MM_CTM_SIMULATION_MAX_VIS_AREA
- MD9616: $MM_CTM_TEACH_HANDW_FEED_P_MIN
- MD9617: $MM_CTM_TEACH_HANDW_FEED_P_REV
- MD9620: $MM_CTM_CYCLE_SAFETY_CLEARANCE
- MD9633: $MM_CTM_INC_DEC_FEED_PER_MIN
- MD9634: $MM_CTM_INC_DEC_FEED_PER_ROT
- MD9637: $MM_CTM_MAX_INP_FEED_P_MIN
- MD9638: $MM_CTM_MAX_INP_FEED_P_ROT
- MD9639: $MM_CTM_MAX_TOOL_WEAR
- MD9648: $MM_CTM_ROUGH_O_RELEASE_DIST
- MD9649: $MM_CTM_ROUGH_I_RELEASE_DIST
- MD10240: $MN_SCALING_SYSTEM_IS_METRIC
- MD20150 [12]: $MC_GCODE_RESET_VALUES
- Tool data: Length X, length Z, wear length radius X and Z, vconst.
- Zero offsets: Position in X, Z.

Program Continuation: Internal

111430 Program not loaded. Error when converting old cycles in G code. No NC memory.

Definitions: In previous ManualTurn versions, LINE, INCLINED and CIRCLE steps were saved as cycles. Now these are saved as G codes (INCLINED, CIRCLE without angle programming).
When loading a sequence, a check is run to see whether old cycles were used. If there are old cycles, the sequence is converted and saved again in the NC. This alarm is output if an error occurs thereby (memory full).

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- -
Remedy: There must be enough memory for the original sequence and for the backup copy in order for the sequence to be created.
Program Continuation: Internal

111900 Start only possible in basic display

Definitions: A G code program can only be started from the basic display of an operating mode (except MANUAL).
Reactions: - Alarm display.
Remedy: Switch to the basic display of an operating mode (except MANUAL). Start single step mode with NC start.
Program Continuation: Internal

111901 Contour is contained in the current program, machining not enabled

Definitions: There is a contour in the current Easystep sequence and it is not permitted to be changed.
Reactions: - Alarm display.
Remedy: Terminate machining. Reload the Easystep sequence and change it correspondingly.
Program Continuation: Internal

111902 Start only with valid reference point

Definitions: The axes have no valid reference point.
Reactions: - Alarm display.
Remedy: Reference all axes.
Program Continuation: Internal

111904 4. Axis not configured, i.e., no driven tool possible

Definitions: 4. The axis is not configured, i.e., no driven tool is possible.
Reactions: - Alarm display.
Remedy: 4. Create an axis. Hereby, the following machine data has to be changed:
- Channel-specific 20070 $MC_AXCONF_MACHAX_USED [3]=4
- Axis-specific for the 4th axis.
- 30300 $MA_IS_ROT_AX=1
- 30310 $MA_ROT_IS_MODULO=1
- 30320 $MA_DISPLAY_IS_ MODULO=1
- 30350 $MA_SIMU_AX_VDI_OUTPUT=1
- 35000 $MA_SPIND_ASSIGN_TO_MACHAX=2

Program Continuation: Internal

112045 Several insertion points required

Definitions: Several insertion points are required for machining the contour pocket. The machining breaks up into several individual machinings.
The program can be started.
This alarm is only a warning.
Residual material will remain.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: By using a smaller milling cutter, the machining could be done with an insertion point.
Program Continuation: Internal

112046 Main contour cannot be bypassed

Definitions: The pocket contour cannot be bypassed with the specified milling cutter.
Residual material will remain.
The program can be started.
This alarm is only a warning.
Reactions: - Alarm display.
Remedy: By using a smaller milling cutter, the pocket contour could be bypassed.
Program Continuation: Internal

112052 No residual material generated

Definitions: No residual material was generated. It could be, that there is no more residual material to machine.
The program can be started.
This alarm is only a warning.
Reactions: - Alarm display.
Remedy: No remedy required.
Program Continuation: Internal

112057 Programmed helix violates contour

Definitions: The starting point for helical insertion was so selected, that the helix violates the programmed contour.
The program can be started.
This alarm is only a warning.
Reactions: - Alarm display.
Remedy: Select another starting point; use a smaller helix radius.
Program Continuation: Internal

112099 System-error contour pocket

Definitions: While the contour pocket was being calculated, an error occurred. The contour pocket cannot be calculated.
The program cannot be started.
Reactions: - Alarm display.
Remedy: Make a note of the error text and contact the Siemens AG, A&D MC hotline.
Program Continuation: Internal

112100 Error during renumbering. Original state restored.

Definitions: The soft key "Renumber" was pressed in the program editor. This caused an error during serial numbering that damaged the program in the memory, making it necessary to reload the initial program into the memory. The program was not renumbered.
Reactions: - Alarm display.
Remedy: Make room in the memory, e.g. by deleting an old program. Select "Renumber" again.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

112200 The contour is a step in the current program sequence. Machining not enabled

Definitions: The contour is an element from a loaded program and cannot be deleted or renamed.
Reactions: - Alarm display.
Remedy: Remove the contour from the loaded program.
Program Continuation: Internal

112201 The contour is a step in the current automatic sequence. Machining not enabled

Definitions: The contour is an element of a program loaded under "Machine auto" and cannot be deleted or renamed.
After the program has been started, the integrated contours cannot be changed under "Program" while the program is running.
Reactions: - Alarm display.
Remedy: Stop the program and load it under "Program". Remove the contour from the program.
Program Continuation: Internal

112210 Tool axis cannot be switched over. Insufficient NC memory.

Definitions: If the tool axis is reselected, the NC program has to be generated again. Hereby the old NC program is saved first.
Then the new program is generated. Here, the NC memory is insufficient to save the new program.
The reselection of the tool axis was not carried out.
Reactions: - Alarm display.
Remedy: Free memory space must be created on the NC, and it must be enough for the program to be processed (e.g. by deleting programs that are no longer required).
Program Continuation: Internal

112211 Tool preselection could not be processed. Insufficient NC memory.

Definitions: When tool preselection is processed, the NC program has to be generated again. Hereby the old NC program is saved first.
Then the new program is generated. Here, the NC memory is insufficient to save the new program.
The tool preselection is not processed.
Reactions: - Alarm display.
Remedy: Free memory space must be created on the NC, and it must be enough for the program to be processed (e.g. by deleting programs that are no longer required).
Program Continuation: Internal

112300 Tool management type 2 not possible. Magazine not completely loaded.

Definitions: The magazine is not completely loaded with tools.
In the magazine of tool management type 2, the number of tools specified in machine data 18082 has to be created.
Reactions: - Alarm display.
Remedy: Installation and start-up: Create the correct number of tools.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

112301 Tool management type 2 not possible. The magazine is not sorted as in the tool list.

Definitions: The magazine list sorting does not correspond to that of the tool list.
In the magazine of tool management type 2, the sequence of the tools has to be defined according to their T numbers.
Reactions: - Alarm display.
Remedy: Installation and start-up: Assign the tools according to their T numbers to the magazine locations.
Program Continuation: Internal

112323 Replace swivel head.

Definitions: The operator is prompted to remove the specified swivel head from the spindle.
Reactions: - Alarm display.
Remedy: Replace swivel head.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112324 Load swivel head.

Definitions: The operator is prompted to load the specified swivel head into the spindle.
Reactions: - Alarm display.
Remedy: Load swivel head.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112325 Exchange swivel head.

Definitions: The operator is prompted to replace the specified swivel head in the spindle with the new swivel head.
Reactions: - Alarm display.
Remedy: Exchange swivel head.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112326 Set swivel head

Definitions: The operator is prompted to set the swivel head in accordance with the specified data.
Reactions: - Alarm display.
Remedy: Set swivel head.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112327 Angle not in permissible area

Definitions: The programmed machining cannot be carried out with the swivel head.
Reactions: - Alarm display.
Remedy: If necessary, clamp the workpiece differently.
Program Continuation: Internal

112328 Angle adapted to angle grid.

Definitions: Due to the angle grid, the swivel head could not be set exactly to the specified angle.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Machining can be continued with the specified values, but it will not correspond exactly to the programming.
Program Continuation: Internal

112329 Set swivel head/table

Definitions: The operator is prompted to set the swivel head/table in accordance with the specified data.
Reactions: - Alarm display.
Remedy: Set swivel head/table.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112330 Set swivel table.

Definitions: The operator is prompted to set the swivel table in accordance with the specified data.
Reactions: - Alarm display.
Remedy: Set swivel table.
When doing this, please follow the machine manufacturer's instructions.
Program Continuation: Internal

112350 No swivel data set up.

Definitions: There are no swivel data blocks.
Reactions: - Alarm display.
Remedy: Set up swivel data blocks (see /FBSP/, ShopMill Description of Functions)
Program Continuation: Internal

112360 The step was not accepted into the program sequence, as the program is running.

Definitions: The program that you want to change is being executed in the operating mode "Machine auto". You can only change programs that are not being executed in the operating mode "Machine auto".
Reactions: - Alarm display.
Remedy: Stop the program run in the "Machine auto" operating mode.
Program Continuation: Internal

112400 Is not available in the tool management

Definitions: The tool specified in the program does not exist.
Reactions: - Alarm display.
Remedy: The tool must be created before the data is saved.
Program Continuation: Internal

112401 Tool could not be created

Definitions: When reading in the tool data, a tool could not be created.
Reactions: - Alarm display.
Remedy: Check tool management.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

112420 Error during the inch/metric conversion! Check all data!

Definitions: The switchover of the data for the inch/metric conversion was not completed.
This alarm can only appear in the event of hardware defects.

Reactions: - NC Start disable in this channel.
- Alarm display.

Remedy: The following data must be checked:
Display MD's:
- MD9655: $MM_CMM_CYC_PECKING_DIST
- MD9656: $MM_CMM_CYC_DRILL_RELEASE_DIST
- MD9658: $MM_CMM_CYC_MIN_COUNT_PO_TO_RAD
- MD9664: $MM_CMM_MAX_INP_FEED_P_MIN
- MD9665: $MM_CMM_MAX_INP_FEED_P_ROT
- MD9666: $MM_CMM_MAX_INP_FEED_P_TOOTH
- MD9670: $MM_CMM_START_RAD_CONTOUR_POCKET
- MD9752: $MM_CMM_MEASURING_DISTANCE
- MD9753: $MM_CMM_MEAS_DIST_MAN
- MD9754: $MM_CMM_MEAS_DIST_TOOL_LENGTH
- MD9755: $MM_CMM_MEAS_DIST_TOOL_RADIUS
- MD9756: $MM_CMM_MEASURING_FEED
- MD9757: $MM_CMM_FEED_WITH_COLL_CTRL
- MD9758: $MM_CMM_POS_FEED_WITH_COLL_CTRL
- MD9759: $MM_CMM_MAX_CIRC_SPEED_ROT_SP
- MD9761: $MM_CMM_MIN_FEED_ROT_SP
- MD9762: $MM_CMM_MEAS_TOL_ROT_SP
- MD9765: $MM_CMM_T_PROBE_DIAM_LENGTH_MEAS
- MD9766: $MM_CMM_T_PROBE_DIAM_RAD_MEAS
- MD9767: $MM_CMM_T_PROBE_DIST_RAD_MEAS
- MD10240: $MN_SCALING_SYSTEM_IS_METRIC
- MD20150 [12]: $MC_GCODE_RESET_VALUES
- Tool data for various cutting edges D: length Z, radius R, wear length radius Z and R.
- Zero offsets: Basic offset position in X, Y, Z , as well as A, C (if available) zero offset.
- Settings in operating mode MANUAL: Retraction plane, safety clearance.

Program Continuation: Internal

112502 Insufficient memory, abort in line %1

Parameters: %1 = Line number

Definitions: The program cannot interpret a program block with contour programming. Contour not in the directory.
Program is not loaded.
The program cannot interpret a program block with contour programming. Contour not in the directory.

Reactions: - Alarm display.

Remedy: Load contour into the directory.
Load contour into the directory.

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

112504 File faulty or not available: %1

Parameters: %1 = File name
Definitions: The program cannot interpret a program block with contour programming.
Contour not in the directory.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Load contour into the directory.
Program Continuation: Internal

112505 Error when interpreting the contour %1

Parameters: %1 = Contour name
Definitions: Contour is faulty.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Check the contour's machining sequence.
Program Continuation: Internal

112506 Maximum number of contour elements exceeded %1

Definitions: The maximum permissible number of 50 contour elements was exceeded when interpreting the machining sequence of a contour.
Reactions: - Alarm display.
Remedy: Check the contour's machining sequence and, if necessary, edit it.
Program Continuation: Internal

112541 Program cannot be interpreted

Definitions: The program cannot be interpreted as a ShopMill program during loading, as the program header is missing.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: -
Program Continuation: Internal

112604 Connection to the PLC broken off

Definitions: Acknowledgement to the PLC user program, that the connection with the PCU has been broken off.
ShopMill PLC is terminated.
Reactions: - Alarm display.
Remedy: Check the PLC user program.
Program Continuation: Internal

112605 Asynchronous subroutine was not executed

Definitions: The input values could not be correctly processed by the NC.
Reactions: - Alarm display.
Remedy: Perform an NCK reset
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

112650 Unknown PLC error

Definitions: The PLC has announced an error that is unknown in the operator panel.
Reactions: - NC Start disable in this channel.
- Alarm display.
Remedy: Press POWER ON, inform Siemens.
Program Continuation: Internal

112999 Faulty grafic data. Exit graphic and restart

Definitions: More data was generated than can be read from the operator panel.
Stop the graphic.
Reactions: - Alarm display.
Remedy: Deselect the graphic and then select it again.
Program Continuation: Internal

120000 Area %1 cannot be loaded! Acknowledge alarm, press area switchover key!

Parameters: %1 = Operating area name
Definitions: One of the applications listed in the REGIE.INI could not be started.
Reactions: - Alarm display.
Remedy: Check whether the entry in REGIE.INI is correct.
Program Continuation: Internal

120001 Area %1 cannot be selected. Please deactivate area %2

Parameters: %1 = Operating area name
%2 = Operating area name
Definitions: Within the scope of an area switchover, a different area is to be terminated (unloaded).
However, the area refuses this.
The area switchover does not take place.
Reactions: - Alarm display.
Remedy: Try again and, if possible, close the reluctant area first.
Program Continuation: Internal

120002 Area %1 is still active. Please deactivate area %1

Parameters: %1 = Operating area name
Definitions: When the MMC system is closed (closing the master control), an area is to be terminated.
However, the area refuses this.
The system was NOT closed.
Reactions: - Alarm display.
Remedy: Try again and, if possible, close the reluctant area first.
Program Continuation: Internal

120003 Area %1 cannot be deactivated. Please try again

Parameters: %1 = Operating area name
Definitions: Within the scope of an area switchover, an area is to be deselected.
However, the area refuses this.
The area switchover does not take place.
Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Try again and, if possible, close the reluctant area first.
Program Continuation: Internal

120005 Please acknowledge the dialog box in area %1

Parameters: %1 = Operating area name
Definitions: The area %1 could not be deselected, as in this area a dialog box is still open.
Reactions: - Alarm display.
Remedy: Close the dialog box in area %1!
Program Continuation: Internal

120006 The channel switchover is currently disabled by area %1.

Parameters: %1 = Operating area name
Definitions: The area %1 has disabled the channel switchover at the moment, as it is performing a critical operation (e.g. execution from external sources, etc.), during which no channel switchover may occur.
Reactions: - Alarm display.
Remedy: Wait until the critical operation is finished or end the critical operation manually.
Program Continuation: Internal

120007 The channel switchover is currently disabled.

Definitions: The channel switchover is currently disabled, as a critical operation, during which no channel switchover may occur, is being carried out.
Reactions: - Alarm display.
Remedy: Wait until the critical operation is finished or end the critical operation manually.
Program Continuation: Internal

120008 Control unit switchover, PLC timeout: 001 control unit switchover, PLC timeout: 002 control unit switchover, PLC timeout: 003

Definitions:
- 001: MMC would like to go offline from this NCU. MMC has made the offline request in the online PLC and is waiting for the positive / negative acknowledgement from the PLC.
- 002: MMC would like to go online to this NCU. MMC has called the target PLC and is waiting for the release to go online.
- 003: MMC has requested the active operating mode and is waiting for acknowledgement from the PLC.

Reactions: - -
Remedy: Check whether the switchover blocks are loaded and started in the online PLC.
Program Continuation: Internal

120120 The alarm list is full of alarm texts: Number of alarm texts too high Alarm texts: File %1 not found alarm texts: Input/Output error in file %1 alarm texts: Input/Output error alarm texts: Error when reading from the index file alarm texts: Error when writing in the index file alarm texts: Syntax error in alarm text file %1

Parameters: %1 = File name
Definitions: Alarm text: Alarm list is full.
- Pending alarms/messages could not be entered into the alarm list due to lack of space. The alarm cannot be deleted, as this event has made the alarm list permanently inconsistent.

Alarm text: Number of alarm texts too high.

© Siemens AG, 2002. All rights reserved

• The number of alarm texts is currently limited to 5000. This limit has been exceeded by the alarm text configuration.

All other alarm texts:

• Alarm texts are stored in files. One of these files could not be accessed properly.

Reactions: - Alarm display.

Remedy:
• Expand the alarm list (Enter maximum number in the file mbdde.ini in the section [Alarms]). Then perform a cold restart for the operator panel.
•
• Reduce the number of alarm texts. Then perform a cold restart for the operator panel.
•
• Make sure that the MMC memory on the hard disk is available after booting, or re-install the MMC software. When entering your own alarm texts, check whether the path and file name are entered correctly in mbdde.ini.

Program Continuation: Internal

120200 Image preparation suppressed

Definitions: The control is so heavily loaded by the processing of a subroutine, that it cannot keep all the display values up-to-date.

Reactions: - Alarm display.

Remedy: The alarm disappears automatically as soon as the overload situation has been eliminated.

If this alarm occurs often, the start-up engineer will have to take appropriate measures (e.g. reduce IPO clock pulse rate).

Program Continuation: Internal

120201 Communication failure

Definitions: The operator panel is connected with the NC and PLC via a serial bus.

This alarm occurs when the communication to these components is interrupted.

In connection with this alarm, all display values connected with the NC/PLC become invalid.

Such faults are normal while the controls are ramping up (e.g. after resetting).

Reactions: - Alarm display.

Remedy: The alarm disappears automatically as soon as the fault situation has ended.

If this alarm is continuously present, a wide variety of faults may be the cause. (e.g. wire breakage, NC/PLC not ramped up, faulty address/data transfer rate configuration of one of the bus nodes, etc.).

Program Continuation: Internal

120202 Waiting for a connection to the NC

Definitions: The operator panel is connected with the NC and PLC via a serial bus.

This alarm occurs if the MMC is started for the first time and the NC/PLC ramp-up is not yet finished or the communication to these components is faulty.

In connection with this alarm, all display values connected with the NC/PLC become invalid.

Such faults are normal while the controls are ramping up (e.g. after resetting).

Reactions: - Alarm display.

Remedy: The alarm disappears automatically as soon as the fault situation has ended.

If this alarm is continuously present, a wide variety of faults may be the cause. (e.g. wire breakage, NC/PLC not ramped up, faulty address/data transfer rate configuration of one of the bus nodes, etc.).

© Siemens AG, 2002. All rights reserved

Program Continuation: Internal

120301 Faulty entry for hardkey 'Program' in Keys.ini.

Definitions: The configuration in Keys.ini is wrong.
Reactions: - Alarm display.
Remedy: In Keys.ini, the parameter ChildTask:=26 has to be set in the line KEY2.0=.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

120302 The selection is not possible. A program has to have been edited first via the area 'Program'.

Definitions: A program can only be selected via the hardkey program if a program has already been edited in the program area.
Reactions: - Alarm display.
Remedy: The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

120303 The selection is not possible. The edited file %1 no longer exists.

Parameters: %1 = Program name with path
Definitions: The file edited last in the program area has in the meantime been deleted.
Reactions: - Alarm display.
Remedy: The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

120304 The selection is not possible. The file %1 has insufficient read rights.

Parameters: %1 = Program name with path
Definitions: The file has insufficient read rights for the current access level.
Reactions: - Alarm display.
Remedy: Set the required read rights by means of keyswitch or password entry.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

120305 Selection is not possible. The file %1 is currently being edited.

Parameters: %1 = Program name with path
Definitions: The file is currently open in another application (e.g. services) with an editor.
Reactions: - Alarm display.
Remedy: Change the program in the already open editor.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

120306

The selection is not possible. The file %1 is selected and active in channel %2.

Parameters: %1 = Program name with path
%2 = Channel number

Definitions: -

Reactions: - Alarm display.

Remedy: Stop the program with the NCU's channel reset and make the selection again.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.

Program Continuation: Internal

120307

The file %1 cannot be opened for the editor because it is selected in channel %2 for execution from external sources.

Parameters: %1 = Program name with path
%2 = Channel number

Definitions: -

Reactions: - Alarm display.

Remedy: A different program on the NCU or for execution from external sources has to be selected.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.

Program Continuation: Internal

120308

In the event of an emergency stop, the program %1 can only be changed in the machine/program correction area.

Parameters: %1 = Program name with path

Definitions: -

Reactions: - Alarm display.

Remedy: Switch to the machine area and change the program with the program correction function.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.

Program Continuation: Internal

120309

The selection is not possible. Please close the simulation and repeat the selection.

Definitions: The simulation is currently active in the program area.
A simultaneous editing is not possible.

Reactions: - Alarm display.

Remedy: Close the simulation and make the selection again.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.

Program Continuation: Internal

120310

The selection is not possible. Please wait for the pending action or terminate it, then repeat the selection.

Definitions: In the program area, programs are currently being copied, loaded or unloaded.
A simultaneous editing is not possible.

Reactions: - Alarm display.

© Siemens AG, 2002. All rights reserved

Remedy: Wait until the action is completed or terminate the action via the soft key "Cancel" and then repeat the selection.
The alarm disappears automatically as soon as a program is edited or simulated in the program area.
The alarm can also be acknowledged manually via diagnostics.
Program Continuation: Internal

300000 Hardware drive bus: DCM not present

Definitions: The DCM (Drive Communication Master, an ASIC on the NCU module that takes control of the drive bus) has not signaled when powering up the drive. In the 840D, a hardware fault is the possible cause of this error. (For the FM-NC, an incorrect configuration is also possible via the NCK MD 13010 DRIVE_LOGIC_NR).
Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Exchange the NCU module.
Program Continuation: Switch control OFF - ON.

300001 Axis %1 drive number %2 not possible

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: On powering up the drives, the NCK-specific machine data 13010 DRIVE_LOGIC_NR was checked for impermissible inputs. In the MD, a drive logic number is entered that can be selected as required within the established limits (drive number 0 = "No drive available"). Numbers greater than 15 are not allowed, nor may the same number be used more than once. The MD array must be configured without spaces, i.e. as soon as the logical drive number 0 is selected once, it is necessary for the logical drive number 0 to be entered in all MDs with a higher location index [p].
Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check MD DRIVE_LOGIC_NR for numbers greater than 15 or for gaps in the machine axis indices.
Program Continuation: Switch control OFF - ON.

300002 Axis %1 drive number %2 assigned twice

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: The drive logic number in the NCK MD 13010 DRIVE_LOGIC_NR has been assigned more than once.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check MD 13010 DRIVE_LOGIC_NR for identical drive logic numbers and assign another number in the range between 0 and 15 (0 corresponds to "No drive available" and is the only number that may occur more than once in the MD array) to each drive.

Program Continuation: Switch control OFF - ON.

300003 Axis %1 drive %2 wrong module type %3

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Incorrect module type

Definitions: The hardware configuration of the drive components established at the time of bus initialization does not correspond to the information in machine data 13030 DRIVE_MODULE_TYPE[p]=... (p ... rack location index).

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enter MD 13030 DRIVE_MODULE_TYPE to match the drive modules that are plugged in.
Select MD 13010 DRIVE_LOGIC_NR and search for the drive logic number that is indicated in the alarm text. The location index number + 1 results in the associated rack location number. The VDD module belonging to this location is determined by the configuration in MD 13030 DRIVE_MODULE_TYPE for the same location index.
Input value 1: 1-axis module, input value 2: 2-axis module.

Program Continuation: Switch control OFF - ON.

300004 Axis %1 drive %2 wrong drive type %3 (FDD/MSD)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Drive type code

Definitions: A feed module is inserted in the rack location determined by the drive logic number, but in the corresponding NCK-specific MD 13040 DRIVE_TYPE, a main spindle is defined (or vice versa).

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
Select MD 13010 DRIVE_LOGIC_NR and search for the drive logic number that is indicated in the alarm text. The location index number + 1 results in the associated rack location number. The FDD/MSD module belonging to this location is determined by the configuration in MD 13040 DRIVE_TYPE for the same location index.
FDD: identifier 1, MSD: identifier 2.

Program Continuation: Switch control OFF - ON.

300005 At least one module found on drive bus that has not been configured

Definitions: At bus initialization at least one module was detected which did not have a drive number, which amounts to one too many. Since all (!) modules on the drive bus must be correctly initialized, all modules therefore also have to be accordingly specified in the machine data.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check machine data; with the NCK MD 13000 DRIVE_IS_ACTIVE a drive that is not yet in use but exists on the bus can be declared as inactive. Inactive drives do not need installation and start-up or drive data.

Program Continuation: Switch control OFF - ON.

300006 Module with drive number %1 has not been found on drive bus

Parameters: %1 = Drive number

Definitions: Not all of the drives stated in MD $MN_DRIVE_LOGIC_NR could be found on the drive bus. You can find the associated module in the configuration display via the displayed drive number.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Perform the following checks:
1) Using the configuration display or the associated machine data, check whether the number and type (1-axis or 2-axis) of the modules match your bus setup.
2) Check whether the red LED is illuminated on the displayed drive module. If this is not the case, then usually the module does not have any power.
- Check the connections of the ribbon cable running from your I/RF or monitoring unit to the module.
- If after switching on the I/RF or monitoring unit, no LED of a module which is connected to it is illuminated, then check the I/RF or monitoring unit and, if required, replace the ribbon cable.
- With a multi-tier installation where the power is switched on at different times, an error message can also mean that one tier has been switched on too late (current permissible time 10 seconds). If possible, switch on the second tier at the same time.

© Siemens AG, 2002. All rights reserved

3) Check whether all drive bus connectors have correctly snapped into place and that the bus terminator is connected.

4) If you have not been able to detect an error by now, the module is defective.

- Replace the module.

Program Continuation: Switch control OFF - ON.

300007 Axis %1 drive %2 not present or inactive

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: In the axis-specific machine data 30110 CTRLOUT_MODULE_NR (this specifies at which drive module the speed setpoint is output) and MD 30220 ENC_MODULE_NR (this specifies the drive module which outputs the encoder actual value for the position control) there is a drive logic number that does not occur in the NCK MD 13010 DRIVE_LOGIC_NR and the machine data 30240 ENC_TYPE and MD 30130 CTRLOUT_TYPE are set to "1".

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the setpoint/actual value assignment in the axis-specific MD 30110 CTRLOUT_ MODULE_NR and MD 30220 ENC_MODULE_NR and the drive logic number in the NCK MD 13010 DRIVE_LOGIC_NR and bring these into agreement.

Program Continuation: Switch control OFF - ON.

300008 Axis %1 drive %2 measuring circuit %3 is not available

Parameters: %1 = NC axis number

%2 = Drive number

%3 = Measuring circuit number

Definitions: In the axis-specific MD 30230 ENC_INPUT_NR [e]=E (e ... encoder index - the position control works with this encoder, E ... encoder number - encoder connector selection on the drive module), an encoder connector (1 or 2) was selected, to which no encoder is connected.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Configure MD 30230 ENC_INPUT_NR [e] according to the encoder connector used or plug the encoder cable onto the other connector.

If the encoder used corresponds to the input in the MD, there is a hardware fault on the drive module. Replace the module!

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300009 Axis %1 drive %2 measuring circuit %3 wrong measuring circuit type (type %4 used)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Measuring circuit number
%4 = Measuring circuit type

Definitions: The available, displayed actual value module on the drive FBG cannot process the signal type that was stated via the axis-specific machine data 30240 ENC_TYPE [e]=S (e ... encoder index - the position control works with this encoder, E ... , S ... signal type of the actual value encoder - 0 ... simulation axis without hardware, 1 ... raw encoder signals, 2 ... rectangle signals).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Set MD 30240 ENC_TYPE [e] to 1 (0 should only be entered for pure simulation axes which are to travel in the actual-value display only).

Program Continuation: Switch control OFF - ON.

300010 Axis %1 drive %2 active without NC axis assignment

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: A drive is active that is not used/addressed by any NC axis (actual value, setpoint).
All active drives must be assigned to an axis with respect to the setpoint value or the actual value.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the configuration data, the assignment of setpoints and actual values for the drive motor and the position encoder.
MDs for the drive configuration:
- Modify MD 13000: DRIVE_IS_ACTIVE
- Modify MD 13010: DRIVE_LOGIC_NR
- Modify MD 13020: DRIVE_INVERTER_CODE
- Modify MD 13030: DRIVE_MODULE_TYPE
- Modify MD 13040: DRIVE_TYPE

MDs for the setpoint/actual-value assignment:
- Modify MD 30110: CTRLOUT_MODULE_NR
- Modify MD 30130: CTRLOUT_TYPE
- Modify MD 30220: ENC_MODULE_NR
- Modify MD 30230: ENC_INPUT_NR
- Modify MD 30240: ENC_TYPE

© Siemens AG, 2002. All rights reserved

It might be necessary to first declare an NC axis in the channel for this drive (MD 20070 AXCONF_MACHAX_USED = K, [K ...channel axis no.]).

Program Continuation: Switch control OFF - ON.

300011 Axis %1 drive %2 hardware version of spindle not supported

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: An old spindle power section (so-called 186-HSA) is connected to the drive bus. This spindle drive is not supported by SINUMERIK 840D. Ramp-up is interrupted.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. A DSP spindle module must be ordered and fitted.

Program Continuation: Switch control OFF - ON.

300012 Axis %1 drive %2 hardware version of control module not supported

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: There is a drive module with an "old" control module on the drive bus. 810D does not support these modules. Ramp-up is interrupted.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Order standard or performance control module and exchange with "old".

Program Continuation: Switch control OFF - ON.

300020 Drive %1 removed for diagnosis

Parameters: %1 = Drive number

Definitions: The alarm indicates that the drive bus configuration has been changed temporarily. The alarm is output if MD 13030 $MN_DRIVE_MODULE_TYPE has value 0 (zero) when a drive is configured.

Reactions:
- Mode group not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Normal operation (full bus configuration): Integrate drive module into the bus again, enter the correct type in MD 13030 $MN_DRIVE_MODULE_TYPE.
- Normal operation (module remains removed): Remove the module in the configuration screen. Remove all connections to outputs and inputs.
- MD30110 $MA_CTRLOUT_MODULE_NR
- MD30130 $MA_CTRLOUT_TYPE

© Siemens AG, 2002. All rights reserved

- MD30220 $MA_ENC_MODULE_NR
- MD30240 $MA_ENC_TYPE
- MD11342 $MA_ENC_HANDWHEEL_MODULE_NR

Program Continuation: Switch control OFF - ON.

300100 Drive power failure

Definitions: In one or several (all) drive modules, there is a power failure although power was previously available. (The timeout is checked for write/read accesses. Timeouts are interpreted as power failure because this is the most probable case. The test takes place in the cyclic mode only but not at system power-up.)

Since the drives in the SINUMERIK 840D and the NC-CPU have the same power supply, this error does not occur here because then the NCU is also without power supply. In the SINUMERIK FM-NC, this error can arise because the power supply is separate.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Switch off the power to the system and switch on again - the drives start up again.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300101 Bus communications failure

Definitions: This alarm indicates that there is still no power supply to the drives although the NC is already running. This message comes only if no drive module has been signaled. (In theory, it could also be a bus error interrupting the connection to the 1st module).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: 1) If the NCU has its own power supply, then the drives still have no power.

- If possible, switch on the power supply for the drives at the same time as the NCU.

2) If the NCU and the drives have the same power supply then not even the first module could be recognized. Check whether the red LED on the first drive module is illuminated. If this is not the case, then usually the module does not have any power.

- Check the connections of the ribbon cable running from your I/RF or monitoring unit to the module.
- If after switching on the I/RF or monitoring unit, no LED of a module which is connected to it is illuminated, then check the I/RF or monitoring unit and, if required, replace the ribbon cable.

3) Check whether all drive bus connectors have correctly snapped into place and that the bus terminator is connected.

4) If you have not been able to detect an error by now, the module is defective.

- Replace the module.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

300200 Drive bus hardware fault

Definitions: The drive bus has a fault. The following causes are possible:
- The bus terminator is missing.
- The drive bus is physically interrupted at some point.
- Miscellaneous hardware fault.

A check line is tested that runs over the entire bus and returns from the last rack location (bus terminator) back to the NCK.

Note: If the drive ramps up correctly even though this message appeared, the error existed only at the beginning of the initialization. In spite of this, the drives can be capable of functioning.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Check the bus terminator.
- Check all plug connections from the drive bus to the drive modules.
- Other hardware faults.

If the remedial measures given above do not lead to a change in the behaviour when starting, please contact the system support for the A&D MC products of SIEMENS AG through the Hotline (tel.: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300201 Axis %1 drive %2 timeout during access, error location %3

Parameters:
%1 = NC axis number
%2 = Drive number
%3 = Error location

Definitions: The read cycle of a drive address in the initialization phase or in cyclic operation has not ended within the monitoring time (approx. 1 ms) (timeout error).

The error can occur in conjunction with a power failure of one or several drive modules. A hardware fault might also be the cause (ASICs, bus, drive modules).

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy: Please inform the authorized personnel/service department. If the alarm has occurred in conjunction with a power failure, this cause of failure must be eliminated. Otherwise, please contact the system support for A&D MC products, SIEMENS AG, through the Hotline (tel.: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300202 Axis %1 drive %2 CRC error, error location %3

Parameters:
%1 = NC axis number
%2 = Drive number
%3 = Error location

© Siemens AG, 2002. All rights reserved

Definitions: The cross-check (CRC) has detected an access error in a write/read cycle. All bus accesses are not controlled directly by the processor but they are handled by special ASICs. They transfer not only the required data but also cross-checks for the write/read data and the addresses.

The error can occur in conjunction with a power failure of one or several drive modules. A hardware fault might also be the cause (ASICs, bus, drive modules).

Reactions:
- NC not ready.
- Mode group not ready, also effective for single axes
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.
- Axes of this channel must be re-referenced.

Remedy: Please inform the authorized personnel/service department. If the alarm has occurred in conjunction with a power failure, this cause of failure must be eliminated. Otherwise, please contact the system support for A&D MC products, SIEMENS AG, through the Hotline (tel.: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300300 Axis %1 drive %2 boot error, error code %3

Parameters:
%1 = NC axis number
%2 = Drive number
%3 = Error code

Definitions: Error occurred while starting up the displayed drive. (Example: Drive signals timeout).

Meaning of the error code:
- 0..5: Timeout while waiting for the acknowledgement from the drive in the displayed state
- 10: No signal from drive CPU (possibly defective module)

Safety Integrated special case: If the axial machine data $MA_SAFE_FUNCTION_ENABLE of at least one axis is not zero, then the occurrence of this alarm with error code 5 can mean that the PLC, after the timeout PLC_RUNNINGUP_TIMEOUT, has not started the cyclic operation. Synchronization of the drive and the PLC is required because in cyclic operation of the drive, data transmission between the PLC and the drive is monitored.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. You can try to power the system up again. The search for the precise cause of error can only be performed by the development team. The displayed status code is always needed for this.

(contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300400 Axis %1 drive %2 system error, error codes %3, %4

Parameters:
%1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

© Siemens AG, 2002. All rights reserved

Definitions: An internal software error or serious error condition has occurred which may be recoverable by hardware reset. Troubleshooting can generally be performed only by Siemens AG, System Support for A&D MC Products, Hotline (tel.: see alarm 1000).

In the error code combination (324,26), the calculation time allocation for the drive communication subtask should be increased via the MD 10140 $MN_TIME_LIMIT_NETTO_DRIVE_TASK (possible up to 500 ms).

If the above-mentioned limit is exhausted and the alarm continues to occur, the MD 10150 $MN_PREP_DRIVE_TASK_CYCLE_RATIO=1 can be set additionally. Please note that by reducing MD 10150, the time share of the preparation in the non-cyclic time plane is reduced. This may lead to longer block cycle times.

Reactions:
- NC not ready.
- Channel not ready.
- If the axis is a single axis when this alarm is triggered, the alarm is only effective for this axis (not effective for e.g. the channel or mode group)
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. You can try to power the system up again. The search for the precise cause of error can only be performed by the development team. The displayed error codes are always needed for this.

(contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300401 Drive software for type %1, block %2 missing or incorrect

Parameters: %1 = Drive type

%2 = Block number

Definitions: Either there is no software for this drive type or it contains errors.

Drive type
- 1 = VSA (as in MD DRIVE_TYPE!)
- 2 = HSA
- 3 = SLM
- 4 = HYD
- 5 = ANA

Block number
- 1 = Drive software (code)
- 2 = Data descriptions (ACC file)

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the data carrier (Jeida board), replace if necessary.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

300402 System error in drive link. Error codes %1, %2

Parameters: %1 = Error code 1

%2 = Error code 2

Definitions: An internal software error or serious error condition has occurred which may be recoverable by hardware reset. Troubleshooting can generally be performed only by Siemens AG, System Support for A&D MC Products, Hotline (tel.: see alarm 1000).

In the error code combination (324,26), the calculation time allocation for the drive communication subtask should be increased via the MD 10140 $MN_TIME_LIMIT_NETTO_DRIVE_TASK (possible up to 500 ms).

If the above-mentioned limit is exhausted and the alarm continues to occur, the MD 10150 $MN_PREP_DRIVE_TASK_CYCLE_RATIO=1 can be set additionally. Please note that by reducing MD 10150, the time share of the preparation in the non-cyclic time plane is reduced. This may lead to longer block cycle times.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Make a note of the error text and contact Siemens AG A&D MC, Hotline (tel./fax: see alarm 1000).

Program Continuation: Switch control OFF - ON.

300403 Axis %1 drive %2 drive software and drive MD with different version numbers

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The version number of the drive software (FDD/MSD) must correspond to the version number stored in the drive machine data because the MD files for different software versions are not compatible.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. After exchanging the drive software, the drives must be installed and started up again. Any MD files that were saved by the control running under the old version must no longer be used.
The old data can be saved with the installation and start-up tool and this data can also be used again.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300404 Axis %1 drive %2 drive MD contains different drive number

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: In the drive MD file loaded in a drive there is a drive number which does not correspond to this drive.

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Files with drive data for a particular drive number must not be copied to another drive.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300405 Axis %1 drive %2 unknown drive alarm, code %3

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Service number

Definitions: The service number signaled by the drive is not implemented in the NCK. It cannot be assigned to any alarm number.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reinitialization of the drive required.
The search for the precise cause of error can only be performed by the development team. The displayed error codes are always needed for this. (contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.: see alarm 1000).

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300410 Axis %1 drive %2 error when storing a file (%3, %4)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Definitions: An attempt to save a data block, i.e. the result of a measuring function, in the file system has failed.
On error code 1 == 291: An error occurred during preparation of the ACC information. Basic information prepared on the drive contains an error or has an unknown format.
On error code 1 == 292: Memory shortage during preparation of the ACC information.

Reactions: - Alarm display.
- Interface signals are set.

Remedy:
- Please inform the authorized personnel/service department.
- Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself (this will require a complete data backup).
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of

© Siemens AG, 2002. All rights reserved

- 18270 $MM_NUM_SUBDIR_PER_DIR,
- 18310 $MM_NUM_DIR_IN_FILESYSTEM,
- Power On
- Reload saved data
- On error code 1 == 291: Replace the drive software and use version with suitable ACC basic information.
- On error code 1 == 292: Replace the drive software and use fewer different versions of the drive software.

Program Continuation: Clear alarm with the RESET key. Restart part program

300411 Axis %1 drive %2 error when reading a file (%3, %4)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Definitions: An attempt to read a data block, e.g. a drive boot file, from the file system has failed. The data block or the file system is damaged.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: If the error occurred during power-up, i.e. it is probably connected to a drive boot file, delete all boot files and load them back into the control from the back-up copy.

Program Continuation: Clear alarm with the RESET key. Restart part program

300412 Error when storing a file (%1, %2)

Parameters: %1 = Error code 1
%2 = Error code 2

Definitions: An attempt to save a data block, i.e. the result of a measuring function, in the file system has failed.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself. To do so, proceed as follows:

- Save all data
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of
- 18270 $MM_NUM_SUBDIR_PER_DIR
- 18310 $MM_NUM_DIR_IN_FILESYSTEM
- Power On
- Reload saved data

Program Continuation: Clear alarm with the RESET key. Restart part program

300413 Error when reading a file (%1, %2)

Parameters: %1 = Error code 1
%2 = Error code 2

© Siemens AG, 2002. All rights reserved

Definitions: An attempt to read a data block, e.g. a drive boot file, from the file system has failed. The data block or the file system is damaged.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: If the error occurred during power-up, i.e. it is probably connected to a drive boot file, delete all boot files and load them back into the control from the back-up copy.

Program Continuation: Clear alarm with the RESET key. Restart part program

300423 Measuring results could not be read (%1)

Parameters: %1 = Error code

Definitions: An attempt to read a measurement result has failed:
- Error code = 4: Not enough space for test result
- Error code = 16: Measurement not yet finished

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Repeat measurement. Alter measuring time if necessary.

Program Continuation: Clear alarm with the RESET key. Restart part program

300500 Axis %1 drive %2 system error, error codes %3, %4

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Definitions: The drive has signaled a system error.
Safety Integrated:
Request: In the corresponding cycle.
On FDD: Generator stop (corresponds to STOP B)
On MSD: Pulse and servo disable (corresponds to STOP A)
The error occurs if the computation time of the drive processor is not sufficient for the cycle indicated in the additional information.
Error no.: 03, additional information: 40, monitoring cycle too small for SINUMERIK Safety Integrated.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Notes on the error codes can be found under Section "Error codes of alarm 300500" in the SINUMERIK 840D/840Di/810D Diagnostics Guide.
NCK reset (POWER ON)
The search for the precise cause of error can only be performed by the development team. The displayed error codes are always needed for this. Reinitialization of the drive required.
Please inform the authorized personnel/service department. (contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.: see alarm 1000)
Safety Integrated: Increase the corresponding cycle or the cascade cycle (e.g. current, speed, position control cycle) or deselect the functions which are not required.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300501 Axis %1 drive %2 maximum current monitoring

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: 1. A serious fault occured during actual current measurement.
2. The maximum current threshold was exceeded while the rotor position identification was active (FDD only).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Modify MD 1107: $MD_INVERTER_MAX_CURRENT (transistor limit current)
- Check the motor data (motor code)
- Check the controller data, current/speed controller gain too high
- Reduce MD 1145: $MD_STALL_TORQUE_REDUCTION (stall torque reduction factor).
- Check the power section and motor terminals (including motor protection), possibly power section too small
- Increase MD 1254: $MD_CURRENT_MONITOR_FILTER_TIME (time constant for current monitoring)
- Error in actual current measurement (if necessary, replace 611D power section or controller module)
- If rotor position identification is active, check MD 1019 $MD_CURRENT_ROTORPOS_IDENT (current for rotor position identification) and, if necessary, reduce the setting.

Program Continuation: Switch control OFF - ON.

300502 Axis %1 drive %2 maximum current monitoring of phase current R

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The phase current R is greater than or equal to 1.05 times the maximum power section current MD 1107: $MD_INVERTER_MAX_CURRENT (transistor limit current).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Check max. power section current in MD 1107 $MD_INVERTER_MAX_CURRENT (transistor limit current)
- Check the motor data (motor code)
- Motor has ground or winding fault
- Check the controller data
- Check the power section and motor terminals (including motor protection)
- Error in actual current measurement (if necessary, replace 611D power section or controller module)

© Siemens AG, 2002. All rights reserved

Program Continuation: Switch control OFF - ON.

300503 Axis %1 drive %2 maximum current monitoring of phase current S

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The phase current S is greater than or equal to 1.05 times the maximum power section current MD 1107: $MD_INVERTER_MAX_CURRENT (transistor limit current).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Check max. power section current in MD 1107 $MD_INVERTER_MAX_CURRENT (transistor limit current)
- Check the motor data (motor code)
- Check the controller data
- Motor has ground or winding fault
- Check the power section and motor terminals (including motor protection)
- Error in actual current measurement (if necessary, replace 611D power section or controller module)

Program Continuation: Switch control OFF - ON.

300504 Axis %1 drive %2 measuring circuit error of motor measuring system

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Signal level of the motor encoder too low or disturbed.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Check encoders, encoder lines and connectors between drive motor and 611D module.
- Check for temporary interruptions (loose contact) caused, for example, by movements in trailing cable.
- Check the shield connection of the front plate of the closed-loop control module (top screw).
- Use original, preassembled encoder cable from Siemens (high degree of shielding).
- If necessary, replace the motor, encoder and/or cables.
- With linear motor, check the signal level; possibly, the measuring scale of the open measuring system is polluted.
- For a gearwheel encoder, check the distance between the gearwheel and the sensor. Replace the sensor or the defective gearwheel.
- Replace the control module.
- Check the metallized intermediate circuit cover.

© Siemens AG, 2002. All rights reserved

Program Continuation: Switch control OFF - ON.

300505 Axis %1 drive %2 measuring circuit error of absolute track, code %3

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Fine error coding

Definitions:
- Incremental encoder (ERN 1387)
- The absolute motor track (C/D track) is monitored for wire-breaks.
- Absolute encoder (EQN 1325)
- Monitoring of the encoder hardware and the EnDat interface
- Accurate diagnostics via error code MD 1023 $MD_ENC_ABS_DIAGNOSIS_MOTOR (diagnostics for measuring system absolute track) on motor measuring systems or MD 1033 $MD_ENC_ABS_DIAGNOSIS_DIRECT (diagnostics for direct measuring system absolute track) for direct measuring systems:
- Overview of bit nos., significance, note:
- Bit 0 Lighting failed
- Bit 1 Signal amplitude too small
- Bit 2 Code connection error
- Bit 3 Overvoltage
- Bit 4 Undervoltage
- Bit 5 Overcurrent
- Bit 6 Battery change necessary
- Bit 7 CRC error (evaluate bit 13) see below, SW 4.2 and higher, synchronous linear motor
- Bit 8 Encoder cannot be used, Assignment of absolute track to incremental track not allowed, SW 4.2 and higher, synchronous linear motor
- Bit 9 C/D track for ERN1387 encoder incorrect (see below) or EQN encoder connected
- Bit 10 Log cannot be aborted
- Bit 11 SSI level detected in data cable
- Bit 12 TIMEOUT while reading measured value
- Bit 13 CRC error
- Bit 14 Wrong IPU submodule for direct measuring signal, only with 611D expansion
- Bit 15 Encoder defective
- CRC error bit 7 and bit 13:
- Bit 7: 0, bit 13: 1 CRC error from SIDA-ASIC
- Bit 7: 1, bit 13: 0 Control check byte error
- Bit 7: 1, bit 13: 1 Error on correction of absolute track by incremental track
- Bits 12 and 15: Zero level monitoring SSI
- Bits 14 and 15: Idle level monitoring SSI
- Note on bit 9:
- Incorrect parameterization (e.g. not on EQN MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM) or MD 1030: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM)
- or old hardware (not suitable for EQN)
- or no encoder connected
- or incorrect encoder cable (for ERN instead of EQN)

© Siemens AG, 2002. All rights reserved

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Check encoders, encoder lines and connectors between drive motor and 611D module. Check for temporary interruptions (loose contact) caused, for example, by movements in trailing cable. If necessary, replace the motor cable.
- Incorrect cable type
- Closed-loop control module defective or not suitable for EnDat interface (e.g. closed-loop control module with EPROM)

Program Continuation: Switch control OFF - ON.

300506 Axis %1 drive %2 NC sign-of-life failure

Parameters:
%1 = NC axis number
%2 = Drive number

Definitions:
Upon servo enable, the NC must update the sign-of-life monitoring in each position control cycle. In case of error, sign-of-life monitoring has not been updated.

Cause:
- a) NC no longer updates the sign-of-life as a reaction to an alarm (e.g. 611D alarm)
- b) Fault occurred during communication via the drive bus
- c) Hardware error on the drive module
- d) NC fault
- e) For 840D: Value of the machine data MD10082: $MN_CTRLOUT_LEAD_TIME (Offset of the setpoint acceptance instant) is too great
- f) MD 1003 $MD_STS_CONFIG not set correctly (e.g. zero)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
Please inform the authorized personnel/service department.
- for a) Determine whether the sign-of-life monitoring failure is a sequential fault. A sequential fault arises, e.g. through: Fault/alarm from axis x with an n-axis configuration. If this fault profile arises, the above-stated error message will be issued for all n-axes, although there is only a fault/alarm at axis x. ==> Remedy the error at axis x ==> sign of life of the other axes is irrelevant.
- for b) Check cable connection, perform remedial measures (check shielding or ground connection).
- for c) Change controller module.
- for d) See NC Diagnostics Guide and change NC hardware if necessary.

© Siemens AG, 2002. All rights reserved

- for e) Correctly set the machine data 840D MD10082: $MN_CTRLOUT_LEAD_TIME (Offset of the setpoint acceptance instant) with the machine data MD10083: $MN_CTRLOUT_LEAD_TIME_MAX (Maximum settable offset of the setpoint acceptance instant).
- for f) Check MD 1300 $MD_STS_CONFIG.

Program Continuation: Switch control OFF - ON.

300507 Axis %1 drive %2 synchronization error of rotor position

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: There is a difference of more than 45° electrical between the present rotor position (C/D track) and the new rotor position as determined by fine synchronization. Faults may have occurred in the encoder or zero marker signals.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Use original Siemens encoder cables (they have a higher degree of screening).
- Check the encoder, encoder cables and screen connection for damage.
- Check the shield connection on the front plate of the controller module (top screw).
- Replace the control module.
- Check MD 1016 $MD_COMMUTATION_ANGLE_OFFSET (commutation angle offset).

Program Continuation: Switch control OFF - ON.

300508 Axis %1 drive %2 zero mark monitoring of motor measuring system

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: An error was detected in the number of encoder lines counted to modulo (16/10) on crossing of the zero marker. Increments were lost or added.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Use original Siemens encoder cables (they have a higher degree of screening).
- Check the encoder, encoder cable and shield connection for loose contact or cable breakage.
- Check the shield connection on the front plate of the controller module (top screw).
- For a gearwheel encoder, check the distance between the gearwheel and the sensor.

© Siemens AG, 2002. All rights reserved

- Replace the encoder, encoder cable or control module.
- Check the metallized intermediate circuit cover.
- If a BERO proximity switch is used, the zero marker of the encoder is still being monitored and not the BERO signal.

Program Continuation: Switch control OFF - ON.

300509 Axis %1 drive %2 current frequency exceeded

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The motor has exceeded the maximum current frequency fmax (see below).
Current frequency = speed * number of motor pole pairs
Maximum current frequency:

- FDD: fmax = 1.12 * ninimum(1.2*MD 1400, MD 1147) * MD1112 / 60
- MSD: fmax = 1.12 * ninimum(MD 1146, MD 1147) * number of pole pairs / 60
- Number of pole pairs = integer component of rated motor frequency (MD 1134) * 60 / rated motor speed (MD 1400)

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Check number of encoder lines in MD 1005: $MD_ENC_RESOL_MOTOR (encoder resolution for motor measuring system).
- Check encoder power supply (short circuit or ground fault).
- Replace encoder, encoder cable or controller module.
- Modify MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed),
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).
- Modify MD 1147: $MD_SPEED_LIMIT (speed limit),
- Modify MD 1112: $MD_NUM_POLE_PAIRS (number of motor pole pairs) (FDD),
- Modify MD 1134: $MD_MOTOR_NOMINAL_FREQUENCY (rated motor frequency) (MSD).

Program Continuation: Switch control OFF - ON.

300510 Axis %1 drive %2 error on actual current measurement zero balancing

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The value of the actual current exceeded the maximum permissible limits during current zero balancing (performed on every pulse disable). For example, the synchronous motor is rotating at a small intermediate circuit voltage and current is flowing across the free-wheeling diodes in the intermediate circuit.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy:
- Error in actual current measurement (if necessary, replace 611D power section or controller module)
- Incorrect power section module (1/2 axis)
- Check the contact between the controller module and the power section
- Check the contact between the fixing screw and the controller module

Program Continuation: Switch control OFF - ON.

300511 Axis %1 drive %2 measuring function active

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The measuring function (e.g. frequency response measurement was active during the switching on operation (power supply start-up active). Illegal activation of the measuring function may have occurred internally.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Stop the measuring function
- NCK reset

Program Continuation: Switch control OFF - ON.

300515 Axis %1 drive %2 power section heat sink temperature exceeded

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The temperature of the power section is acquired from a temperature switch on the heatsink. 20 seconds after the heatsink temperature warning, drive shutdown is initiated immediately in order to avoid thermal damage to the power section (regenerative stop).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Provide better ventilation of the drive modules, e.g. by means of:
- Greater air throughput in the switching cabinet, if necessary cool the ambient air of the 611D modules.
- Avoid numerous acceleration and deceleration operations in rapid sequence by modifying the workpiece programming.
- Incorrect motor/power section dimensioning
- Excessive ambient temperature (see Planning Guide)
- Exceeding of the installation height (see Planning Guide)
- Excessive pulse frequency (see Planning Guide)
- Defective module

© Siemens AG, 2002. All rights reserved

- Fan failure
- Observance of the minimum clearance over and under the power section (see Planning Guide)

Program Continuation: Switch control OFF - ON.

300604 Axis %1 drive %2 motor encoder is not adjusted

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The stored serial number of the encoder on a linear motor encoder with an EnDat interface does not match the number of the active encoder. It is therefore assumed that the encoder has not been started up with the motor in question before or has not been adapted to the motor.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: For 1FN3 linear motors: Measure the rotor position offset to the EMF of the U_R phase and add it as commutation angle offset to MD 1016 $MD_COMMUTATION_ANGLE_OFFSET. Then set MD 1017 $STARTUP_ASSISTANCE to "-1" to store the serial number of the Endat encoder. Then save the bootfiles and execute an NCK Reset. Otherwise: To determine the commutation angle offset in MD 1016, initiate the rotor position identifier via MD 1017 = 1. After the error has been acknowledged, the identification is carried out.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300605 Axis %1 drive %2 motor change not valid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: An attempt was made to switch over to a motor data record that is not parameterized.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Parameterize the motor data record selected or switch over to another motor.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300606 Axis %1 drive %2 flux controller at limit

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The specified flux setpoint cannot be achieved even though the maximum current has been provided.

Causes:
- Incorrect motor data (replacement circuit diagram data)
- Motor data and motor connection type (star/delta) do not match

© Siemens AG, 2002. All rights reserved

- Motor has become unstable because motor data grossly wrong
- Current limit is too low for the motor (0.9 * MD 1238 * MD 1103 < MD 1136)
- Power section too small

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Remedy the cause.
- Modify MD 1238: $MD_CURRENT_LIMIT (current limit)
- Modify MD 1103: $MD_MOTOR_NOMINAL_CURRENT (motor nominal current)
- Modify MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current)
- Use greater power section.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300607 Axis %1 drive %2 current controller at limit

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The specified current setpoint cannot be injected into the motor even though the maximum voltage has been provided. Cause: Motor not connected or phase missing.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Check the motor converter connection (phase missing).
- Check the motor protection.
- DC link voltage available?
- Check the DC link connections (check that the screws are tight).
- Check the contact between the closed-loop control module and the power section.
- Check the contact between the fixing screw and the closed-loop control module.
- The Uce monitoring circuit has been activated (perform a reset by switching the power supply off and on again).
- Replace the control module.
- Replace the power section.
- Replace the motor.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300608 Axis %1 drive %2 speed controller at limit

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The speed controller output is lying for an impermissibly long time at its limit (MD 1605: $MD_SPEEDCTRL_LIMIT_TIME and MD 1606: The torque setpoint has exceeded the torque limit or the current setpoint the current limit. The monitoring system is only active when the speed setpoint is below the speed threshold in MD 1606: $MD_SPEEDCTRL_LIMIT_THRESHOLD.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Is the motor blocked, overloaded or the brake closed?
- If permitted by the power section, set the limits for torque, performance and current to higher values.
- Is the motor connected to ground?
- Check the motor converter connection (phase missing, incorrect rotary field).
- Check the encoder resolution.
- Check the encoder, encoder cable and shield connection for loose contact or cable breakage.
- Check the direction of rotation of the encoder tracks (e.g. gearwheel encoder MD 1011: $MD_ACTUAL_VALUE_CONFIG bit 1)
- Is the encoder cable appropriate for the encoder type?
- Check the controller settings (e.g. after software exchange).
- Check the motor protection.
- DC link voltage available?
- Check the DC link connections (check that the screws are tight).
- The Uce monitoring circuit has been activated (perform a reset by switching the power supply off and on again).
- Modify MD 1605: $MD_SPEEDCTRL_LIMIT_TIME and MD 1606: $MD_SPEEDCTRL_LIMIT_THRESHOLD in accordance with the mechanical and dynamic features of the axis.
- Default values for FDD:
- MD 1605 = 200 ms
- MD 1606 = 8000 rpm
- Default values for MSD:
- MD 1605 = 200 ms
- MD 1606 = 30 rpm
- Replace the motor (encoder is defective, motor has a winding or ground fault or a short circuit)
- With linear motors:
- Check actual value inversion.
- Check the reduction of the max. motor current MD 1105 MD_MOTOR_MAX_CURRENT_REDUCTION and increase the value if necessary.
- Check connection of motor cables.
- Are the motors arranged correctly in a parallel circuit and is their electrical interconnection correct?

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

© Siemens AG, 2002. All rights reserved

300609 Axis %1 drive %2 encoder cut-off frequency exceeded

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Actual speed value exceeds encoder limit frequency fg,max = 650kHz; fg = nist * MD 1005

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- The wrong encoder may be in use.
- Correct MD 1005: Does the number of encoder lines match the setting in MD 1005 $MD_ENC_RESOL_MOTOR (encoder resolution for motor measuring system)?
- Is the motor encoder cable connected correctly?
- Is the motor encoder cable shield installed flat?
- Replace the motor (the encoder is defective).
- Check the encoder, encoder cable and shield connection for loose contact or cable breakage.
- Replace the encoder.
- Replace the 611D control module.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300610 Axis %1 drive %2 rotor position identification failed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The rotor position could not be determined from the measurement signals. For detailed diagnostics see also: MD1734: $MD_DIAG_ROTORPOS_IDENT.

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Increase MD1019: $MD_CURRENT_ROTORPOS_IDENT (current rotor position identification) and use greater power section, if necessary.
- Check armature inductance MD1116: $MD_ARMATURE_INDUCTANCE and increase if necessary.
- Check the motor converter connection (phase missing).
- Check the motor protection.
- DC link voltage available?
- Check the DC link connections (check that the screws are tight).
- The Uce monitoring circuit has been activated (perform a reset by switching the power supply off and on again).
- Replace the 611D power section.
- Replace the control module.

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300611 Axis %1 drive %2 generator mode: Motion at rotor position identification

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: During the measurement, the motor has turned by more than the permissible value entered in MD1020: $MD_MAX_TURN_ROTORPOS_IDENT (maximum rotation of the rotor position identification). The rotation can be caused by switching on a rotating motor or by the identification itself.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- If the rotation was caused by the identification itself, and if the error occurs repeatedly, then reduce MD1019: $MD_CURRENT_ROTORPOS_IDENT or increase MD1020: $MD_MAX_TURN_ROTORPOS_IDENT (maximum rotation of the rotor position identification).
- Brake the motor during the identification.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300612 Axis %1 drive %2 illegal current during rotor position identification

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: 1. With rotor position identification active, the current was
>= 1.2 * 1.05 * INVERTER_MAX_CURRENT (MD 1107)
2. With rotor position definition active, the current was
>= MOTOR_MAX_CURRENT (MD 1104)

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check MD 1019 with rotor position identification active and, if necessary, reduce the setting.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300613 Axis %1 drive %2 maximum permissible motor temperature exceeded

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The motor temperature (measured via the temperature sensor KTY 84 and fed to the module through the motor encoder cable) has exceeded the temperature limit in drive MD 1607: $MD_MOTOR_TEMP_SHUTDOWN_LIMIT (motor temperature shutdown limit).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Motor overloaded.
- Check the motor data. Possibly the machine current was too high as a result of incorrect motor data.
- Check the temperature sensor (2nd sensor possible with MSD).
- Check the motor encoder cable.
- Motor encoder defective.
- Check the motor fan.
- Check the acceleration.
- If necessary, use a higher-performance motor.
- Winding fault in motor.
- 611D control module defective.
- Modify MD 1230: $MD_TORQUE_LIMIT (1st torque limit), MD 1235: POWER_LIMIT (1st power limit) set too high.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300614 Axis %1 drive %2 time monitoring of motor temperature

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The motor temperature (measured via the temperature sensor KTY 84 and fed to the module through the motor encoder cable) has exceeded the temperature limit in drive MD 1602: $MD_MOTOR_TEMP_WARN_LIMIT for a longer period of time than permitted in drive machine data 1603 $MD_MOTOR_TEMP_ALARM_TIME. $MD_MOTOR_TEMP_ALARM_TIME (time stage motor temperature alarm).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Motor overloaded.
- Check the motor data. Possibly the machine current was too high as a result of incorrect motor data.
- Check the temperature sensor.
- Check the motor encoder cable.

© Siemens AG, 2002. All rights reserved

- Check the motor fan.
- Motor encoder defective.
- Check the acceleration.
- If necessary, use a higher-performance motor.
- Winding fault in motor.
- 611D control module defective.
- Modify MD 1230: $MD_TORQUE_LIMIT (1st torque limit), MD 1235: POWER_LIMIT (1st power limit) set too high.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

300701 Axis %1 drive %2 start-up required

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: This alarm appears when installing and starting up for the first time without valid 611D machine data!

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Reset motor data.
- Back up boot drive.
- Repeat the Power ON.

Program Continuation: Switch control OFF - ON.

300702 Axis %1 drive %2 base cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The base cycle time set on the NC was too high for the drive.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: 840D: NCK RESET. After powering up the system again, the NCK machine data 10 050: $MN_SYSCLOCK_CYCLE_TIME (system base cycle) and MD 10080: $MD_SYSCLOCK_SAMPL_TIME_RATIO (division factor of position control cycle for actual value acquisition) are also modified automatically so that the limits are respected.
840C: Change the base cycle on the NC in MD 168.

Program Continuation: Switch control OFF - ON.

300703 Axis %1 drive %2 current cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: An illegal value has been entered in the drive MD 1000: $MD_CURRCTRL_CYCLE_TIME.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
The following values are permissible:
611D control module Current controller cycle
- Standard control as 1-axis: >= 125 µs
- Standard control as 2-axis: >= 125 µs
- Performance 1-axis: >= 62.5 µs
- Performance 2-axis: >= 125 µs
- 810D: >= 156.25 µs
- MCU: >= 125 µs
- Performance 2: >= 31.25 µs

Program Continuation: Switch control OFF - ON.

300704 Axis %1 drive %2 speed controller cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: An illegal value has been entered in the drive MD 1001: $MD_SPEEDCTRL_CYCLE_TIME.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
The following values are permissible:
611D control module Speed controller cycle
- Standard control as 1-axis: >= 125 µs
- Standard control as 2-axis: >= 500 µs
- Performance 1-axis: >= 62.5 µs (MSD >= 125 µs)
- Performance 2-axis: >= 125 µs
- 810D: >= 312.5 µs
- MCU: >= 125 µs
- Performance 2, 1-axis: >= 31.25 µs
- Performance 2, 2-axis: >= 62.5 µs

Program Continuation: Switch control OFF - ON.

300705 Axis %1 drive %2 position controller cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The monitor in the 611D module has detected a position controller pulse rate which is beyond the permissible limits. The conditions for a permissible position controller pulse rate are:

© Siemens AG, 2002. All rights reserved

1. Minimum cycle period: 250µs (810D 312.5µs)
2. Maximum pulse rate: 4 s
3. The position controller pulse rate must be a multiple of the speed controller cycle given in the drive MD 1001: $MD_SPEEDCTRL_CYCLE_TIME.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Change the position controller pulse rate on the NC.

Program Continuation: Switch control OFF - ON.

300706 Axis %1 drive %2 monitoring cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Monitoring cycle MD 1002: $MD_MONITOR_CYCLE_TIME is invalid.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. See Drive Functions "FB/ DB1".

Program Continuation: Switch control OFF - ON.

300707 Axis %1 drive %2 basic cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: In a 2-axis module, the basic drive cycle for the two axes is not the same. This alarm can only occur with OEM users who have the 611D drive without the standard NCK interface. This makes it possible for axially different basic drive cycles to be transferred to the 611D modules.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set the same basic drive cycle for both axes.

Program Continuation: Switch control OFF - ON.

300708 Axis %1 drive %2 current controller cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The current controller cycle MD 1000: $MD_CURRCTRL_CYCLE_TIME must be identical for both axes on 2-axis modules.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The current controller cycle MD 1000: $MD_CURRCTRL_CYCLE_TIME must be identical for both axes on 2-axis modules.

Program Continuation: Switch control OFF - ON.

300709 Axis %1 drive %2 speed controller cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The speed controller cycle MD 1001: $MD_SPEEDCTRL_CYCLE_TIME must be identical for both axes on 2-axis modules.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reduce speed controller cycle MD 1001: $MD_SPEEDCTRL_CYCLE_TIME must be identical for both axes on 2-axis modules.

Program Continuation: Switch control OFF - ON.

300710 Axis %1 drive %2 position controller cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: In a 2-axis module, the position controller cycle for the two axes is not the same. This alarm can only occur with OEM users having the 611D drives without the standard NCK interface. This would make it possible to transfer axially different position controller cycles to the 611D module.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set an identical position controller cycle for both axes.

Program Continuation: Switch control OFF - ON.

300711 Axis %1 drive %2 monitoring cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The monitoring cycle MD 1002: $MD_MONITOR_CYCLE_TIME must be identical for both axes on 2-axis modules.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Modify MD 1002: $MD_MONITOR_CYCLE_TIME for both axes.

Program Continuation: Switch control OFF - ON.

300712 Axis %1 drive %2 configuration of controller structure (higher dynamic response) not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: On a 2-axis module, an attempt was made to change the control structure via MD 1004: $MD_CTRL_CONFIG in such a way that speed control would be performed in advance of current control. This is only allowed on 1-axis modules in order to improve the dynamic response!

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. In the drive MD 1004: $MD_CTRL_CONFIG bit 2 must be set to zero (no) (default setting). This ensures that the current control acts before the speed control.

Program Continuation: Switch control OFF - ON.

300713 Axis %1 drive %2 lead time for position controller invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The derivative-action time describes the point in time the setpoints are taken over in the drive. The derivative-action defined by the NC must be smaller than the position controller cycle. The derivative-action time must be an integer multiple of the speed controller cycle.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Modify MD 10082: $MN_CTROUT_LEAD_TIME (derivative-action time).

Program Continuation: Switch control OFF - ON.

300714 Axis %1 drive %2 power section code invalid

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The code number of the power section entered in drive MD 1106: $MD_INVERTER_CODE (power section code number) does not match the power section in the drive configuration display (MD 13020: $MC_DRIVE_INVERTER_CODE).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Delete modified machine data (e.g. controller data). Reset the drive (delete the bootfile) and repeat the startup.

Program Continuation: Switch control OFF - ON.

300715 Axis %1 drive %2 maximum power section current less than or equal to zero

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The maximum current of the power section entered in drive MD 1107: $MD_INVERTER_MAX_CURRENT (transistor limit current) is less than or equal to zero.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reset the drive (delete the bootfile) and repeat the startup.

Program Continuation: Switch control OFF - ON.

300716 Axis %1 drive %2 torque constant less than or equal to zero

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: 1. The value in drive MD 1113: $MD_TORQUE_CURRENT_RATIO (torque constant) is less than or equal to zero.

2. The ratio of MD1113: $MD_TORQUE_CURRENT_RATIO (torque constant) / MD1112: $MD_NUM_POLE_PAIRS (motor pole pairs) is greater than 70.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

For standard motors: Reset the drive (delete the bootfile) and repeat the startup.

For third-party motors: Enter a valid value in drive MD 1113: $MD_TORQUE_CURRENT_RATIO (torque constant), or check and, if necessary, correct the ratio of MD1113: $MD_TORQUE_CURRENT_RATIO (torque constant) / MD1112: $MD_NUM_POLE_PAIRS (motor pole pairs).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300717 Axis %1 drive %2 motor moment of inertia less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1117: $MD_MOTOR_INERTIA (motor moment of inertia) is less than or equal to zero.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup. For MSD, configure "1st motor" first.
For third-party motors: Enter a valid value in drive MD 1117: $MD_MOTOR_INERTIA (motor moment of inertia).

Program Continuation: Switch control OFF - ON.

300718 Axis %1 drive %2 calculation dead time of current controller less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1101: $MD_CTRLOUT_DELAY (dead time of current control circuit) is less than or equal to zero. The dead time is calculated internally and automatically initialized according to the type of module (1/2-axis, standard/performance module, 810D).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reset the drive (delete the bootfile) and repeat the startup. Check drive MD 1101: $MD_CTRLOUT_DELAY (dead time of current control circuit).

Program Continuation: Switch control OFF - ON.

300719 Axis %1 drive %2 motor not parameterized for delta operation

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: On activating the star/delta switchover by drive MD 1013: $MD_ENABLE_STAR_DELTA, the motor delta (motor 2) is not parameterized.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and enter the machine data for motor delta (motor 2).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300720 Axis %1 drive %2 maximum motor speed invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Because of the high maximum motor speed in the drive MD 1401: $MD_MOTOR_MAX_SPEED and the speed controller cycle in MD 1001: $MD_SPEEDCTRL_CYCLE_TIME sufficiently high speeds can occur to cause a format overflow. Example: A motor speed of 480,000 rpm can still be processed without error at a speed controller cycle time of 125 µs.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reduce the maximum motor speed MD 1401: $MD_MOTOR_MAX_SPEED (speed for the maximum useful motor speed) or set a smaller speed controller cycle in MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle).

Program Continuation: Switch control OFF - ON.

300721 Axis %1 drive %2 zero-load current greater than rated motor current

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The no-load current of the motor (MD 1136: $MD_MOTOR_NOLOAD_CURRENT) has been set at a greater value than the rated current of the motor (MD 1103: $MD_MOTOR_NOMINAL_CURRENT).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors:
Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors:
Check and, if necessary, refer to the motor data sheet to correct machine data MD 1103: $MD_MOTOR_NOMINAL_CURRENT (rated motor current) and MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).

Program Continuation: Switch control OFF - ON.

300722 Axis %1 drive %2 zero-load motor current greater than rated current of power section

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: On the basis of its no-load current (MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current), the connected motor is too large for the power section in use (continuous thermal current MD 1108: $MD_INVERTER_MAX_THERMAL_CORR (current limit for power section).

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

1. Reset the drive (delete the bootfile) and repeat the startup.

2. Check the configuration and install a suitable power section for the motor. Repeat the startup.

Program Continuation: Switch control OFF - ON.

300723 Axis %1 drive %2 STS configuration of axes differ

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The configuration of the control block MD 1003: $MD_STS_CONFIG (STS configuration) must be identical for both axes on 2-axis modules.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check drive MD 1003: $MD_STS_CONFIG (STS configuration) and set the bits for the two axes of the module so that they are the same. (Do not change the default setting - this corresponds to the optimum configuration).

Program Continuation: Switch control OFF - ON.

300724 Axis %1 drive %2 number of pole pairs invalid

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: FDD: The configured number of pole pairs in the drive MD 1112: $MD_NUM_POLE_PAIRS is outside the permissible range limits.

MSD: Modify MD 1134: $MD_MOTOR_NOMINAL_FREQUENCY (rated motor frequency) or MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed) is not correct.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

For standard motors: Reset the drive (delete the bootfile) and repeat the startup.

For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1112: $MD_NUM_POLE_PAIRS (number of motor pole pairs).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300725 Axis %1 drive %2 number of encoder marks of measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The number of encoder marks of the motor measuring system in the drive MD 1005: $MD_ENC_RESOL_MOTOR (number of encoder marks of the motor measuring system) is zero or greater than the maximum input limit.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Match the number of encoder marks of the motor measuring system in drive MD 1005: ENC_RESOL_MOTOR (number of encoder marks of the motor measuring system) to the encoder in use. (Default setting for motor measuring system: (Default setting for motor measuring system: 2048 incr./rev.).

Program Continuation: Switch control OFF - ON.

300726 Axis %1 drive %2 voltage constant is zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in drive MD 1114: $MD_EMF_VOLTAGE is set to zero.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1114: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).

Program Continuation: Switch control OFF - ON.

300727 Axis %1 drive %2 reactance less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance) or MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance) or MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance) is less than or equal to zero.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance) or MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance) or MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance).
Program Continuation: Switch control OFF - ON.

300728 Axis %1 drive %2 adaption factor torque/current too high

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: The matching factor from setpoint torque to cross current in the speed controller is too large.
Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1103: $MD_MOTOR_NOMINAL_CURRENT (rated motor current) or MD 1107: $MD_INVERTER_MAX_CURRENT (transistor limit current) or MD 1113: $MD_TORQUE_CURRENT_RATIO (torque constant).
Program Continuation: Switch control OFF - ON.

300729 Axis %1 drive %2 motor zero-speed current less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: The value in MD 1118: $MD_MOTOR_STANDSTILL_CURRENT is less than or equal to zero.
Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1118: $MD_MOTOR_STANDSTILL_CURRENT (motor standstill current).
Program Continuation: Switch control OFF - ON.

300730 Axis %1 drive %2 rotor resistance invalid

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: The value in drive MD 1138: $MD_ROTOR_COLD_RESISTANCE (cold rotor resistance) is less than or equal to zero or a format overflow has occurred.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: One of the following machine data may contain an invalid value:
- Modify MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle)
- Modify MD 1134: $MD_MOTOR_NOMINAL_FREQUENCY (rated motor frequency)
- Modify MD 1138: $MD_ROTOR_COLD_RESISTANCE (cold rotor resistance)
- Modify MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance)
- Modify MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance)
- Modify MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance)

Fulfill the condition according to the following formula:
16 * MD1001 * 0.00003125 * MD1138 * 2PI * MD1134 / (MD1140 + MD1141) < 1
Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Switch control OFF - ON.

300731 Axis %1 drive %2 rated power less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1130: $MD_MOTOR_NOMINAL_POWER (rated motor power) is less than or equal to zero.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1130: $MD_MOTOR_NOMINAL_POWER (rated motor power).

Program Continuation: Switch control OFF - ON.

300732 Axis %1 drive %2 rated speed less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed) is less than or equal to zero.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed).
Program Continuation: Switch control OFF - ON.

300733 Axis %1 drive %2 zero load voltage invalid

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: Error in the no-load voltage (MD 1135):
- MD 1135 <= 0 or
- MD 1135 > MD 1132 or
- MD 1135 x MD 1142/MD 1400 + Uvor > 450V.

Where
Uvor = 0.181 x MD 1136 x MD 1142 x MD 1119
- Modify MD 1135: $MD_MOTOR_NOLOAD_VOLTAGE (motor no-load voltage)
- Modify MD 1132: $MD_MOTOR_NOMINAL_VOLTAGE (rated motor voltage)
- Modify MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed)
- Modify MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening)
- Modify MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current)
- Modify MD 1119: $MD_SERIES_INDUCTANCE (series reactor inductance)

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors:
- Modify MD 1132: $MD_MOTOR_NOMINAL_VOLTAGE (motor voltage)
- Modify MD 1135: $MD_MOTOR_NOLOAD_VOLTAGE (motor no-load voltage)
- Modify MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed)
- Modify MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening)
- Modify MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).
- Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Switch control OFF - ON.

300734 Axis %1 drive %2 zero load current less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number
Definitions: The value in MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current) is less than or equal to zero.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).

Program Continuation: Switch control OFF - ON.

300735 Axis %1 drive %2 field weakening speed invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening) is less than or equal to zero.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening).

Program Continuation: Switch control OFF - ON.

300736 Axis %1 drive %2 Lh characteristic invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1143: $MD_LH_CURVE_UPPER_SPEED (LH curve upper speed) is less than or equal to the value in MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening) or the value in MD 1144: $MD_LH_CURVE_GAIN (LH curve gain) is less than 100.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors:
- Modify MD 1143: $MD_LH_CURVE_UPPER_SPEED (Lh curve upper speed)
- Modify MD 1144: $MD_LH_CURVE_GAIN (Lh curve gain)

© Siemens AG, 2002. All rights reserved

- Modify MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening) should be checked and, if necessary corrected.
- Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Switch control OFF - ON.

300737 Axis %1 drive %2 configuration of two EnDat encoders not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The MCU hardware does not allow two absolute encoders to be connected on one axis with the EnDat interface.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Disconnect one of the two absolute encoders:
Replace the motor measuring system by using another motor or use another encoder for direct measuring system.
Correct the corresponding entries in MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM) or MD 1030: $MD_ACTUAL_VALUE_CONFIG_DIRECT (actual value sensing configuration DM).

Program Continuation: Switch control OFF - ON.

300738 Axis %1 drive %2 module number for measuring system not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The NC has assigned the direct measuring system to an axis which doesn't have a motor measuring system. This error can only occur with the 810D!

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the configuration of the direct measuring system. See NC-MD 30220: $MA_ENC_MODULE_NR and NC-MD 30230: $MA_ENC_INPUT_NR.

Program Continuation: Switch control OFF - ON.

300739 Axis %1 drive %2 measuring system already used as motor measuring system

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The NC has assigned the direct measuring system to a measuring system output which is already used by another motor measuring system. This error can only occur with the 810D!

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the configuration of the direct measuring system. See NC-MD 30220: $MA_ENC_MODULE_NR and NC-MD 30230: $MA_ENC_INPUT_NR.

Program Continuation: Switch control OFF - ON.

300740 Axis %1 drive %2 measuring system used several times

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The NC has assigned the direct measuring system to a measuring system output which is already used by another direct measuring system. This error can only occur with the 810D!

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the configuration of the direct measuring system. See NC-MD 30220: $MA_ENC_MODULE_NR and NC-MD 30230: $MA_ENC_INPUT_NR.

Program Continuation: Switch control OFF - ON.

300741 Axis %1 drive %2 asynchronous mode: feedforward control gain out of range

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If motor inertia and motor nominal torque have been selected unfavorably, the asynchronous motor feedforward control gain is beyond the range of the internal number format.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- Operation without encoder: Reduce the number of encoder lines in MD 1005: $MD_ENC_RESOL_MOTOR (encoder resolution for motor measuring system), since this has an impact on the internal number format. Optional/additional measure: see operation with encoder
- Operation with encoder: Reduce speed controller cycle MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300742 Axis %1 drive %2 voltage/frequency mode: converter frequency invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Only converter frequencies (MD 1100: $MD_PWM_FREQUENCY (pulse width modulation frequency)) of 4 kHz or 8 kHz are permissible in V/f mode (selected via MD 1014: $MD_UF_MODE_ENABLE).

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and, if necessary, correct the torque adaptation factor MD 1100: $MD_PWM_FREQUENCY (pulse width modulation frequency) or remedy the problem by deselecting V/f mode MD 1014: $MD_UF_MODE_ENABLE.

Program Continuation: Switch control OFF - ON.

300743 Axis %1 drive %2 function not supported on this 611D controller module

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The 611D Performance control module is required for SINUMERIK Safety Integrated. If this hardware has not been installed, this alarm is triggered. The alarm also occurs if 1PH2/4/6 motors are connected and no 611D performance control module is available. The following function is not supported in connection with 611D Comfort modules: Motor switchover (MD1013 >0) and MD1100 not equal to MD2100.

Safety Integrated: Booting is interrupted, the pulses remain disabled. The 611D Performance control module is required for SINUMERIK Safety Integrated. If this hardware has not been installed, this alarm is triggered. This alarm is also triggered if the motors 1PH2/4/6 are connected and no 611D Performance control module has been installed.

Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Replace the 611D control module.

Safety Integrated: Replace the 611D control module.

Program Continuation: Switch control OFF - ON.

300744 Axis %1 drive %2 safety monitoring checksum invalid, confirmation and acceptance test required!

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The actual checksum of the safety-relevant MDs calculated by the drive and stored in MD 1398: $MD_SAFE_ACT_CHECKSUM (display of the checksum of the machine data for safe functions) has another value than the setpoint checksum stored during the last machine acceptance in MD 1399: $MD_SAFE_DES_CHECKSUM (checksum of the machine data for safe functions). The safety-relevant data have been modified or there is an error.

Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check, and if necessary, correct all safety-relevant MDs. Then perform POWER ON. Perform acceptance test.

Program Continuation: Switch control OFF - ON.

300745 Axis %1 drive %2 limit values for safe end positions exchanged

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: For the safe end position monitoring, there is a smaller value in the MD for the upper limit value than in the MD for the lower limit value.

Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Check the MDs
- Modify MD 1334: $MD_SAFE_POS_LIMIT_PLUS[n] (upper limit value for safe limit position) and
- Modify MD 1335: $MD_SAFE_POS_LIMIT_MINUS[n] (lower limit value for safe limit position)

and modify so that the upper limit value exceeds the lower limit value.

Program Continuation: Switch control OFF - ON.

300746 Axis %1 drive %2 SBH/SG not enabled

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: In the machine data 1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable) the function SBH/SG has not been enabled although the function SE/SN has been selected in this MD.

Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enable the function SBH/SG via MD 1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable).

© Siemens AG, 2002. All rights reserved

Program Continuation: Switch control OFF - ON.

300747 Axis %1 drive %2 monitoring cycle time MD 1300 invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Check and, if necessary, correct MD 1300: $MD_SAFETY_CYCLE_TIME (monitoring cycle) has not been set as a multiple of the NC position control cycle.
Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set the monitoring cycle via MD 1300 to n * NC position control cycle, n must be >= 1.

Program Continuation: Switch control OFF - ON.

300748 Axis %1 drive %2 monitoring cycle times of both axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The monitoring cycles set in MD 1300: $MD_SAFETY_CYCLE_TIME (monitoring cycle) for the two axes of a 2-axis module are not identical.
Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and, if necessary, correct MD 1300: $MD_SAFETY_CYCLE_TIME (monitoring cycle) on all drives of the module.

Program Continuation: Switch control OFF - ON.

300749 Axis %1 drive %2 conversion factor between motor and load too large

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The conversion factor from the motor system [increments] to the load system [µm/mdeg] is larger than 1 or the factor which converts the load system to the motor system is larger than 65535.
Conditions:
The condition for the factor load system to motor system is: µm_to_incr <= 65535
The condition for the factor motor system to load system is: incr_to_µm <= 1
with µm_to_incr = 1 / incr_to_µm
Formula for rotary axis:
The following applies for rotary motor encoder and rotary axis:
incr_to_µm(n) = (MD1321 SAFE_ ENC_ GEAR_ DENOM(n) / (MD1322 SAFE_ ENC_ NUMERA(n)) * incr_to_µm_rot_rotax
with n = 0 ... 7 (gear stage) and
incr_to_µm_rot_rotax = (360000 / 8192) * (1 / MD1318 SAFE_ENC_RESOL)

© Siemens AG, 2002. All rights reserved

- MD 1318 SAFE_ENC_RESOL (number of encoder lines per revolution)
- MD 1321 SAFE_ENC_GEAR_DENOM[n] (encoder/load gear denominator)
- MD 1322 SAFE_ENC_GEAR_NUMERA[n] (encoder/load gear numerator)

Formula for linear axis:

The following applies for rotary motor encoder and linear axis:

incr_to_µm(n) = (MD1321 SAFE_ ENC_ GEAR_ DENOM(n) / (MD1322 SAFE_ ENC_ NUMERA(n)) * incr_to_µm_rot_lin

incr_to_µm_rot_lin = (1000 / 8192) * (1 / MD1318 SAFE_ENC_RESOL) * MD1320 SAFE_ENC_GEAR_PITCH

Explanations:

- MD 1318 SAFE_ENC_RESOL (number of encoder lines per revolution)
- MD 1320 SAFE_ENC_GEAR_PITCH (spindle pitch)
- MD 1321 SAFE_ENC_GEAR_DENOM[n] (encoder/load gear denominator)
- MD 1322 SAFE_ENC_GEAR_NUMERA[n] (encoder/load gear numerator)
- n = 0 ... 7 (gear stage)

Request: During boot-up of the control.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the following safety-relevant MDs depending on the motor encoder type and axis type and correct, if necessary.

- MD 1317 SAFE_ENC_GRID_POINT_DIST Grid division linear scale (for linear encoder)
- MD 1318 SAFE_ENC_RESOL Encoder marks per revolution (for rotary encoder)
- MD 1318 SAFE_ENC_RESOL
- MD 1320 SAFE_ENC_GEAR_PITCH (for rotary encoder and linear axis)
- MD 1321 SAFE_ENC_GEAR_DENOM
- MD 1322 SAFE_ENC_GEAR_NUMERA (when using a gear)
- The motor encoder type and the axis type are determined via MD 1302: $MD_SAFE_IS_ROT_AX.

Program Continuation: Switch control OFF - ON.

300750 Axis %1 drive %2 parameterization error in speed control adaption

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The upper adaptation speed MD 1412: $MD_SPEEDCTRL_ADAPTSPEED_2 (upper adaption speed) is less than the lower adaptation speed MD 1411: $MD_SPEEDCTRL_ADAPTSPEED_1 (lower adaption speed).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. $MD_SPEEDCTRL_ADAPTSPEED_2 (upper adaption speed) and MD 1411: $MD_SPEEDCTRL_ADAPTSPEED_1 (lower adaption speed).

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300751 Axis %1 drive %2 speed control gain too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The proportional gain of the speed controller MD 1407: $MD_SPEEDCTRL_GAIN_1 (P gain of speed controller) or MD 1408: $MD_SPEEDCTRL_GAIN_2 (P gain of upper adaption speed) has been set too high.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set a smaller value for the proportional gain in MD 1407: $MD_SPEEDCTRL_GAIN_1 (P gain of speed controller) or MD 1408: $MD_SPEEDCTRL_GAIN_2 (P gain of upper adaption speed). (Allow for the active speed controller adaptation.)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300752 Axis %1 drive %2 blocking frequency of setpoint current filter too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The blocking frequency of a current setpoint filter is greater than the reciprocal value of 2 current controller cycles (violation of the sampling theorem). (1/2*MD 1000*31.25 microsec)

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The value of machine data MD 1210: $MD_CURRENT_FILTER_1_SUPPR_FREQ (blocking frequency of current setpoint filter 1) or MD 1213: $MD_CURRENT_FILTER_2_SUPPR_FREQ (blocking frequency of current setpoint filter 2) or MD 1216: $MD_CURRENT_FILTER_3_SUPPR_FREQ (blocking frequency of current setpoint filter 3) or MD 1219: $MD_CURRENT_FILTER_4_SUPPR_FREQ (blocking frequency of current setpoint filter 4) must be less than the reciprocal value of two current controller cycles MD 1000: $MD_CURRCTRL_CYCLE_TIME (current controller cycle). (1/2*MD 1000*31.25 microsec)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300753 Axis %1 drive %2 rotor position identification current less than minimal value

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The current set in MD1019: $MD_CURRENT_ROTORPOS_IDENT (rotor position identification current) that is smaller than the minimum value permissible for the motor.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The current set in MD1019: $MD_CURRENT_ROTORPOS_IDENT (rotor position identification current) that is not smaller the permissible minimum value (40% for non-Siemens synchronous linear motor (SLM)).

Possibly, a larger power section must be used.

If allowed with the motor used, hide the error by setting bit 5 in MD 1012: $MD_FUNCTION_SWITCH (function switch).

Notice: Motors with a low saturation response (e.g. 1FN3 linear motors) might react to very low identification currents with misorientation. This can lead to uncontrolled movements.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300754 Axis %1 drive %2 signal number of var. signaling function invalid

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The signal number for the output of the corresponding signaling function is not permissible. The signal number range is between 0 and 25.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enter the correct signal number.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300755 Axis %1 drive %2 voltage/frequency mode: motor is turning

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The motor did not come to a standstill when the voltage/frequency mode was activated.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Stop the motor before activating the voltage/frequency mode.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

300756 Axis %1 drive %2 speed hysteresis of setpoint current smoothing invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1246: $MD_CURRENT_SMOOTH_HYSTERESIS $MD_CURRENT_SMOOTH_HYSTERESIS (hysteresis of the speed-dependent M setpoint smoothing) is greater than or equal to the value in MD 1245: $MD_CURRENT_SMOOTH_HYSTERESIS (threshold of the speed-dependent M setpoint smoothing).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and, if necessary, correct the torque adaptation factor MD 1246: $MD_CURRENT_SMOOTH_HYSTERESIS (hysteresis of the speed-dependent M setpoint smoothing) or MD 1245: $MD_CURRENT_SMOOTH_HYSTERESIS (threshold of the speed-dependent M setpoint smoothing). $MD_CURRENT_SMOOTH_SPEED.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300757 Axis %1 drive %2 adaption factor of torque limit invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The torque adaptation factor MD 1191: $MD_TORQUE_LIMIT_ADAPT_SERVO (adaptation of servo limit torque) exceeds the format limit.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: Check and, if necessary, correct the torque adaptation factor MD 1191: $MD_TORQUE_LIMIT_ADAPT_SERVO (adaptation of servo limit torque).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300758 Axis %1 drive %2 generator mode: response voltage > switch-off threshold

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The sum of the values in MD1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) + MD1632: $MD_LINK_VOLTAGE_GEN_HYST (voltage range for generator control) is greater than MD1633: $MD_LINK_VOLTAGE_GEN_OFF (deactivation threshold for generator axis).

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Modify drive machine data

- Modify MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) or
- Modify MD 1632: $MD_LINK_VOLTAGE_GEN_HYST (voltage range for generator control) or
- Modify MD 1633: $MD_LINK_VOLTAGE_GEN_OFF (voltage range for generator control) or

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300759 Axis %1 drive %2 generator mode: response voltage > monitoring threshold

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The value in MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) is greater than MD1630: $MD_LINK_VOLTAGE_MON_THRESHOLD (response threshold for DC link monitoring).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Modify drive machine data

- Modify MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) or
- Modify MD 1630: $MD_LINK_VOLTAGE_MON_THRESHOLD (response threshold for DC link monitoring).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300760 Axis %1 drive %2 generator mode: emergency retraction speed > max. motor speed

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The value entered for the emergency retraction speed in MD1639: $MD_RETRACT_SPEED (emergency retraction speed) is greater than MD1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Modify drive machine data
- Modify MD 1639: $MD_RETRACT_SPEED (emergency retraction speed) or
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300761 Axis %1 drive %2 generator mode: minimum axis speed > max. motor speed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD1635: $MD_GEN_AXIS_MIN_SPEED $MD_GEN_AXIS_MIN_SPEED (minimum generator axis speed) is greater than the value in MD1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Modify drive machine data
- Modify MD 1635: $MD_GEN_AXIS_MIN_SPEED generator axis) or
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300762 Axis %1 drive %2 emergency retraction mode/generator mode already active

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Emergency retraction or generator mode already active.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check parameterization/ machine data.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

300763 Axis %1 drive %2 emergency retraction mode/generator mode invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Value specified by the NC via a G command must be in the range 0 to 7.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check parameterization (G command in the NC).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300764 Axis %1 drive %2 emergency retraction mode/generator mode not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Emergency retraction/Generator operation is only possible with an active DC link measurement (MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage = 0). In an old hardware version, no DC link measurement is possible and therefore the error message 300765 might appear in addition if MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage) is set to 0 in an old hardware version.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter the value zero in the machine data MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage), or order a new hardware version of the control module.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300765 Axis %1 drive %2 measurement of DC link voltage not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If the fixed voltage MD 1161: $MD_FIXED_LINK_VOLTAGE(DC link fixed voltage) = 0, no DC link measurement is possible because the hardware version is wrong.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department. Enter a value greater than zero in the machine data MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage), or order a new hardware version of the control module.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300766 Axis %1 drive %2 blocking frequency > Shannon frequency

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The blocking frequency of a speed setpoint filter is greater than the Shannon sampling frequency from the sampling theorem.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Current setpoint filter:

The value in MD 1210, 1213,1216 or 1219 must be smaller than the inverse value of two current controller cycles MD 1000: $MD_CURRENTCTRL_CYCLE_TIME (current controller cycle) (1/2*MD 1000 *31.25 microsec).

Speciality with SINUMERIK 810D:

Current setpoint filters 2,3 and 4 are calculated in the speed controller cycle. This means that MD 1213, 1216 or 1219 must be smaller than the inverse value of two speed controller cycles MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle) (1/2*MD 1001 *31.25 microsec).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300767 Axis %1 drive %2 natural frequency > Shannon frequency

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The natural frequency of a speed setpoint filter is greater than the Shannon sampling frequency from the sampling theorem.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

The natural frequency in Hz of a speed setpoint filter must be less than the reciprocal value of two speed controller cycles.

Speed setpoint filter 1:

MD 1520 * 0.01 * MD 1514 < 1 / (2 * MD 1001 * 31.25 microsec)

Speed setpoint filter 2:

MD 1521 * 0.01 * MD 1517 < 1 / (2 * MD 1001 * 31.25 microsec)

© Siemens AG, 2002. All rights reserved

- Modify MD 1520: $MD_SPEED_FILTER_1_BS_FREQ (bandstop filter natural frequency speed setpoint filter 1)
- Modify MD 1514: $MD_SPEED_FILTER_1_SUPPR_FREQ (suppression frequency speed setpoint filter 1) 1)
- Modify MD 1521: $MD_SPEED_FILTER_2_BS_FREQ (bandstop filter natural frequency speed setpoint filter 2)
- Modify MD 1517: $MD_SPEED_FILTER_2_SUPPR_FREQ (suppression frequency speed setpoint filter 2) 2)
- Modify MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300768 Axis %1 drive %2 numerator bandwidth > double blocking frequency

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The bandwidth numerator of a current or speed setpoint filter is greater than twice the blocking frequency.

This error message is only issued for the general bandstop filter if:

- Speed setpoint filter 1:
- MD 1516 > 0.0 or
- MD 1520 <> 100.0
- Speed setpoint filter 2:
- MD 1519 > 0.0 or
- MD 1521 <> 100.0
- Current setpoint filter 1:
- MD 1212 > 0.0
- Current setpoint filter 2:
- MD 1215 > 0.0
- Current setpoint filter 3:
- MD 1218 > 0.0
- Current setpoint filter 4:
- MD 1221 > 0.0

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

The bandwidth numerator must be less than twice the blocking frequency.

- Current setpoint filter 1:
- MD 1212 <= 2 * MD 1210
- Current setpoint filter 2:
- MD 1215 <= 2 * MD 1213
- Current setpoint filter 3:
- MD 1218 <= 2 * MD 1216
- Current setpoint filter 4:
- MD 1221 <= 2 * MD 1219

© Siemens AG, 2002. All rights reserved

- Speed setpoint filter 1:
- MD 1516 <= 2 * MD 1514
- Speed setpoint filter 2:
- MD 1519 <= 2 * MD 1517
- Modify MD 1212: $MD_CURRENT_FILTER_1_BW_NUM (numerator bandwidth current setpoint filter 1)
- Modify MD 1210: $MD_CURRENT_FILTER_1_SUPPR_FREQ (suppression frequency current setpoint filter 1)
- Modify MD 1215: $MD_CURRENT_FILTER_2_BW_NUM (numerator bandwidth current setpoint filter 2)
- Modify MD 1213: $MD_CURRENT_FILTER_2_SUPPR_FREQ (suppression frequency current setpoint filter 2)
- Modify MD 1218: $MD_CURRENT_FILTER_3_BW_NUM (numerator bandwidth current setpoint filter 3)
- Modify MD 1216: $MD_CURRENT_FILTER_3_SUPPR_FREQ (suppression frequency current setpoint filter 3)
- MD1221: $MD_CURRENT_FILTER_4_BW_NUM (numerator bandwidth current setpoint filter 4)
- Modify MD 1219: $MD_CURRENT_FILTER_4_SUPPR_FREQ (suppression frequency current setpoint filter 4)
- Modify MD 1516: $MD_SPEED_FILTER_1_BW_NUMERATOR (numerator bandwidth speed setpoint filter 1)
- Modify MD 1514: $MD_SPEED_FILTER_1_SUPPR_FREQ (suppression frequency speed setpoint filter 1)
- Modify MD 1519: $MD_SPEED_FILTER_2_BW_NUMERATOR (numerator bandwidth speed setpoint filter 2)
- Modify MD 1517: $MD_SPEED_FILTER_2_SUPPR_FREQ (suppression frequency speed setpoint filter 2)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300769 Axis %1 drive %2 denominator bandwidth > double natural frequency

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The bandwidth denominator of a current or speed setpoint filter is greater than twice the natural frequency.

This error message is only issued for the general bandstop filter if:

- Speed setpoint filter 1:
- MD 1516 > 0.0 or
- MD 1520 <> 100.0
- Speed setpoint filter 2:
- MD 1519 > 0.0 or
- MD 1521 <> 100.0
- Current setpoint filter 1:
- MD 1212 > 0.0
- Current setpoint filter 2:
- MD 1215 > 0.0
- Current setpoint filter 3:
- MD 1218 > 0.0
- Current setpoint filter 4:
- MD 1221 > 0.0

© Siemens AG, 2002. All rights reserved

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

The bandwidth denominator of a current or speed setpoint filter must be less than twice the natural frequency.

- Speed setpoint filter 1:
- MD 1515 <= 2 * MD 1514 * 0.01 * MD 1520
- Speed setpoint filter 2:
- MD 1518 <= 2 * MD 1517 * 0.01 * MD 1521
- Current setpoint filter 1:
- MD 1211 <= 2 * MD 1210
- Current setpoint filter 2:
- MD 1214 <= 2 * MD 1213
- Current setpoint filter 3:
- MD 1217 <= 2 * MD 1216
- Current setpoint filter 4:
- MD 1220 <= 2 * MD 1219
- Modify MD 1515: $MD_SPEED_FILTER_1_BANDWIDTH (bandwidth speed setpoint filter 1)
- Modify MD 1514: $MD_SPEED_FILTER_1_SUPPR_FREQ (suppression frequency speed setpoint filter 1)
- Modify MD 1520: $MD_SPEED_FILTER_1_BS_FREQ (bandstop filter natural frequency speed setpoint filter 1)
- Modify MD 1518: $MD_SPEED_FILTER_2_BANDWIDTH (bandwidth speed setpoint filter 2)
- Modify MD 1517: $MD_SPEED_FILTER_2_SUPPR_FREQ (suppression frequency speed setpoint filter 2)
- Modify MD 1521: $MD_SPEED_FILTER_2_BS_FREQ (bandstop filter natural frequency speed setpoint filter 2)
- Modify MD 1211: $MD_CURRENT_FILTER_1_BANDWIDTH (bandwidth current setpoint filter 1)
- Modify MD 1210: $MD_CURRENT_FILTER_1_SUPPR_FREQ (suppression frequency current setpoint filter 1)
- Modify MD 1214: $MD_CURRENT_FILTER_2_BANDWIDTH (bandwidth speed setpoint filter 2)
- Modify MD 1213: $MD_CURRENT_FILTER_2_SUPPR_FREQ (suppression frequency current setpoint filter 2)
- Modify MD 1217: $MD_CURRENT_FILTER_3_BANDWIDTH (bandwidth speed setpoint filter 3)
- Modify MD 1216: $MD_CURRENT_FILTER_3_SUPPR_FREQ (suppression frequency current setpoint filter 3)
- Modify MD 1220: $MD_CURRENT_FILTER_4_BANDWIDTH (bandwidth speed setpoint filter 4)
- Modify MD 1219: $MD_CURRENT_FILTER_4_SUPPR_FREQ (suppression frequency current setpoint filter 4)

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300770 Axis %1 drive %2 format error

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The calculated filter coefficients of a bandstop filter are beyond the range of the internal format.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Change the filter setting. The Hotline provides support for accurate troubleshooting. Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300771 Axis %1 drive %2 asynchronous mode: converter frequency invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Only a converter frequency of 4 kHz or 8 kHz is permitted in asynchronous mode (selected by MD 1465 < MD 1146).
- Modify MD 1465: $MD_SWITCH_SPEED_MSD_AM (switchover speed MSD/AM)
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Modify MD 1100: $MD_PWM_FREQUENCY (pulse width modulation frequency) or deselect AM mode (make the value in MD 1465: $MD_SWITCH_SPEED_MSD_AM less than the value in MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300772 Axis %1 drive %2 asynchronous mode: speed control gain too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1451: $MD_SPEEDCTRL_GAIN_1_AM (proportional gain of AM speed controller) is too high.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The current set in MD1019: $MD_SPEEDCTRL_GAIN_1_AM (proportional gain of AM speed controller).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300773 Axis %1 drive %2 asynchronous mode: feedforward control structure not possible

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: In asynchronous mode (select by MD 1465 < MD 1146) a feedforward control structure (MD 1004, bit 0 = 1) is not possible.

- Modify MD 1465: $MD_SWITCH_SPEED_MSD_AM (switchover speed MSD/AM)
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed)
- Modify MD 1004: $MD_CTRL_CONFIG (configuration structure)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Rectify the error in the entry for the feedforward control structure MD 1004 or by deselecting the asynchronous mode MD 1465 > MD 1146.

- Modify MD 1004: $MD_CTRL_CONFIG (configuration structure)
- Modify MD 1465: $MD_SWITCH_SPEED_MSD_AM (switchover speed MSD/AM)
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300774 Axis %1 drive %2 asynchronous mode: changeover speed invalid

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: In mixed operation MSD/AM (MD 1465 > 0) only closed-loop controlled AM mode is allowed (MD1466 <= MD1465).

- Modify MD 1465: $MD_SWITCH_SPEED_MSD_AM (switchover speed MSD/AM)
- Modify MD 1466: $MD_SWITCH_SPD_OPEN_LOOP_AM (switchover speed closed-loop/open-loop control AM)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

© Siemens AG, 2002. All rights reserved

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Rectify error by selecting pure AM mode (MD1465 = 0) or by deselecting the AM-controlled mode (MD1466 < MD1465).

- Modify MD 1465: $MD_SWITCH_SPEED_MSD_AM (switchover speed MSD/AM)
- Modify MD 1466: $MD_SWITCH_SPD_OPEN_LOOP_AM (switchover speed closed-loop/open-loop control AM)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300775 Axis %1 drive %2 fixed link voltage of axes differ

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: An unequal fixed voltage MD1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage) has been found for axes of a drive module. Since a fixed voltage <> 0 replaces the measured DC link voltage value, but the DC link voltage value is measured only once for all axes of a drive module, the fixed voltage must be identical on all module axes before it is accepted.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set the same fixed voltage (MD 1161) on all module axes. $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage) has been found for axes of a drive module.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300776 Axis %1 drive %2 measuring circuit monitoring must be active

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: On FDD:

The control is disabled, the motor is decelerated, SIMODRIVE_READY and DRIVE_READY are canceled.

On MSD:

Pulse delete, motor coasts, SIMODRIVE_READY and DRIVE_READY are canceled.

Note: The reaction (FDD, MSD) can be configured via 611D-MD 1613.0.

Request: During boot-up of the control and cyclically.

With active Safety Integrated (MD 1301 <> 0: $MD_SAFE_FUNCTION_ENABLE (safe functions enable)), the measuring circuit monitoring of the motor (incremental) must be activated via MD 1600: $MD_ALARM_MASK_POWER_ON (concealable alarms (Power On) bit 4.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Activate measuring circuit monitoring of motor (incremental).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300777 Axis %1 drive %2 rotor position identification current too high

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The current set in MD1019: $MD_CURRENT_ROTORPOS_IDENT (current rotor position identification) is greater than the permissible current for the motor and the power section used.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Reduce MD1019: $MD_CURRENT_ROTORPOS_IDENT (current rotor position identification).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300778 Axis %1 drive %2 generator mode: converter frequency rotor position identification

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: When selecting the rotor position identification (MD1011 bit 12 or bit 13) only converter frequencies (MD1100) of 4 kHz/8 kHz are permissible.

- Modify MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM)
- Modify MD 1100: $MD_PWM_FREQUENZY (pulse width modulation frequency)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

Change the converter frequency (MD 1100) or deselect rotor position identification (MD1011 bit 12 or bit 13).

© Siemens AG, 2002. All rights reserved

- Modify MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM)
- Modify MD 1100: $MD_PWM_FREQUENZY (pulse width modulation frequency)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300779 Axis %1 drive %2 motor moment of inertia less than or equal to zero

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The value in MD 1117: $MD_MOTOR_INERTIA (motor moment of inertia) is less than or equal to zero.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For MSD, configure "1st motor" first.
- For third-party motors: Enter a valid value in drive MD 1117: $MD_MOTOR_INERTIA (motor moment of inertia).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300780 Axis %1 drive %2 zero load current > rated motor current

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The no-load current of the motor (MD 1136: $MD_MOTOR_NOLOAD_CURRENT) has been set at a greater value than the rated current of the motor (MD 1103: $MD_MOTOR_NOMINAL_CURRENT).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: Check and, if necessary, refer to the motor data sheet to correct machine data MD 1103: $MD_MOTOR_NOMINAL_CURRENT (rated motor current) and MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

300781 Axis %1 drive %2 zero load current > rated current of power section

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: On the basis of its no-load current (MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current), the connected motor is too large for the power section in use (continuous thermal current MD 1108: $MD_INVERTER_MAX_THERMAL_CORR (current limit for power section).

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

1. Reset the drive (delete the bootfile) and repeat the startup.

2. Check the configuration and install a suitable power section for the motor. Repeat the startup.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300782 Axis %1 drive %2 reactance less than or equal to zero

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The value in MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance) or MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance) or MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance) is less than or equal to zero.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance) or MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance) or MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300783 Axis %1 drive %2 rotor resistance invalid

Parameters: %1 = NC axis number

%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The value in drive MD 1138: $MD_ROTOR_COLD_RESISTANCE (cold rotor resistance) is less than or equal to zero or a format overflow has occurred.

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.

- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: One of the following machine data may contain an invalid value:
- Modify MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle)
- Modify MD 1134: $MD_MOTOR_NOMINAL_FREQUENZY (rated motor frequency)
- Modify MD 1138: $MD_ROTOR_COLD_RESISTANCE (cold rotor resistance)
- Modify MD 1139: $MD_STATOR_LEAKAGE_REACTANCE (stator leakage reactance)
- Modify MD 1140: $MD_ROTOR_LEAKAGE_REACTANCE (rotor leakage reactance)
- Modify MD 1141: $MD_MAGNETIZING_REACTANCE (magnetizing reactance)

Fulfill the condition according to the following formula:

16 * MD1001 * 0.00003125 * MD1138 * 2PI * MD1134 / (MD1140 + MD1141) < 1

Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300784 Axis %1 drive %2 zero load voltage invalid

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: Error in the no-load voltage (MD 1135):

- MD 1135 <= 0 or
- MD 1135 > MD 1132 or
- MD 1135 x MD 1142/MD 1400 + Uvor > 450V.
- Where
- Uvor = 0.181 x MD 1136 x MD 1142 x MD 1119
- Modify MD 1135: $MD_MOTOR_NOLOAD_VOLTAGE (motor no-load voltage)
- Modify MD 1132: $MD_MOTOR_NOMINAL_VOLTAGE (rated motor voltage)
- Modify MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed)
- Modify MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening)
- Modify MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current)
- Modify MD 1119: $MD_SERIES_INDUCTANCE (series reactor inductance)

The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
For third-party motors: Check and, if necessary, correct the following machine data with reference to the data sheet:
- Modify MD 1132: $MD_MOTOR_NOMINAL_VOLTAGE (motor voltage)
- Modify MD 1135: $MD_MOTOR_NOLOAD_VOLTAGE (motor no-load voltage)
- Modify MD 1400: $MD_MOTOR_RATED_SPEED (rated motor speed)
- Modify MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening)
- Modify MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).
- Call the SIEMENS AG, SIMODRIVE Hotline.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300785 Axis %1 drive %2 zero load current less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current) is less than or equal to zero.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1136: $MD_MOTOR_NOLOAD_CURRENT (motor no-load current).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300786 Axis %1 drive %2 field weakening speed invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening) is less than or equal to zero.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.
- For standard motors: Reset the drive (delete the bootfile) and repeat the startup.
- For third-party motors: Check and, if necessary, refer to the motor data sheet to correct MD 1142: $MD_FIELD_WEAKENING_SPEED (threshold speed for field weakening).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300787 Axis %1 drive %2 asynchronous mode: feedforward control gain out of range

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If motor inertia and motor nominal torque have been selected unfavorably, the asynchronous motor feedforward control gain is beyond the range of the internal number format.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Operation without encoder: Reduce the number of encoder lines in MD 1005: $MD_ENC_RESOL_MOTOR (encoder resolution for motor measuring system), since this has an impact on the internal number format.
- Optional/additional measure: see operation with encoder
- Operation with encoder: Reduce speed controller cycle MD 1001: $MD_SPEEDCTRL_CYCLE_TIME (speed controller cycle).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300788 Axis %1 drive %2 parameterization error in current control adaption

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The upper current limit in MD 1181: $MD_CURRCTRL_ADAPT_CURRENT_2 (upper adaption current limit) is less than the lower current limit in MD 1180: $MD_CURRCTRL_ADAPT_CURRENT_1 (lower adaption current limit).

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Modify MD 1181: $MD_CURRCTRL_ADAPT_CURRENT_2 (upper adaption current limit) and MD 1180: $MD_CURRCTRL_ADAPT_CURRENT_1 (lower adaption current limit).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300789 Axis %1 drive %2 function not supported on this 611D controller module

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: A function was selected that is not possible with this closed-loop control module.
This alarm appears, if:
- a non-available acceleration sensor was activated in MD 1560.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Switch off the non-selectable function or use another closed-loop control module!

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

300799 Axis %1 drive %2 data backup and reboot required

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Since drive machine data have been changed, it is necessary to recalculate parameters. This is initiated by pressing the soft key CALCULATE. After calculating the control parameters, it is necessary to save the machine data and to reboot.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The newly calculated data must be saved (Soft key: SAVE). The new parameters will become effective during the next boot procedure!

Program Continuation: Switch control OFF - ON.

300850 Axis %1 drive %2 parameterization error in speed control adaption

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The upper adaptation speed MD 1412: $MD_SPEEDCTRL_ADAPTSPEED_2 (upper adaptation speed) is less than the lower adaptation speed MD 1411: $MD_SPEEDCTRL_ADAPTSPEED_1 (lower adaptation speed).

Reactions:
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Modify MD 1412: $MD_SPEEDCTRL_ADAPTSPEED_2 (upper adaptation speed) and MD 1411: $MD_SPEEDCTRL_ADAPTSPEED_1 (lower adaptation speed).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300854 Axis %1 drive %2 signal number of var. signaling function invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The signal number for the output of the corresponding signaling function is not permissible. The signal number range is between 0 and 25.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Enter the correct signal number.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300855 Axis %1 drive %2 voltage/frequency mode: motor is turning

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The motor did not come to a standstill when the voltage/frequency mode was activated.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Stop the motor before activating the voltage/frequency mode.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300858 Axis %1 drive %2 generator mode: response voltage > switch-off threshold

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The sum of the values in MD1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) + MD1632: $MD_LINK_VOLTAGE_GEN_HYST (voltage range for generator control) is greater than MD1633: $MD_LINK_VOLTAGE_GEN_OFF (deactivation threshold for generator axis).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Modify drive machine data
- Modify MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) or
- Modify MD 1632: $MD_LINK_VOLTAGE_GEN_HYST (voltage range for generator control) or
- Modify MD 1633: $MD_LINK_VOLTAGE_GEN_OFF (voltage range for generator control) or

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300859 Axis %1 drive %2 generator mode: response voltage > monitoring threshold

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) is greater than MD1630: $MD_LINK_VOLTAGE_MON_THRESHOLD (response threshold for DC link monitoring).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Modify MD 1631: $MD_LINK_VOLTAGE_GEN_ON (response voltage for generator axis) or
- Modify MD 1630: $MD_LINK_VOLTAGE_MON_THRESHOLD (response threshold for DC link monitoring).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

300860 Axis %1 drive %2 generator mode: emergency retraction speed > max. motor speed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value entered for the emergency retraction speed in MD1639: $MD_RETRACT_SPEED (emergency retraction speed) is greater than MD1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Modify MD 1639: $MD_RETRACT_SPEED (emergency retraction speed) or
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300861 Axis %1 drive %2 generator mode: minimum axis speed > max. motor speed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The value in MD1635: $MD_GEN_AXIS_MIN_SPEED $MD_GEN_AXIS_MIN_SPEED (minimum generator axis speed) is greater than the value in MD1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
- Modify MD 1635: $MD_GEN_AXIS_MIN_SPEED generator axis) or
- Modify MD 1146: $MD_MOTOR_MAX_ALLOWED_SPEED (maximum motor speed).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300862 Axis %1 drive %2 emergency retraction mode/generator mode already active

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Emergency retraction or generator mode already active.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check parameterization/ machine data.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300863 Axis %1 drive %2 emergency retraction mode/generator mode invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Value specified by the NC via a G command must be in the range 0 to 7.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check parameterization (G command in the NC).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300864 Axis %1 drive %2 emergency retraction mode/generator mode not possible

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: Emergency retraction/Generator operation is only possible with an active DC link measurement (MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage = 0). In an old hardware version, no DC link measurement is possible and therefore the error message 300765 might appear in addition if MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage) is set to 0 in an old hardware version.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enter the value zero in the machine data MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage), or order a new hardware version of the control module.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300865 Axis %1 drive %2 measurement of DC link voltage not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If the fixed voltage MD 1161: $MD_FIXED_LINK_VOLTAGE(DC link fixed voltage) = 0, no DC link measurement is possible because the hardware version is wrong.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Enter a value greater than zero in the machine data MD 1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage), or order a new hardware version of the control module.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300875 Axis %1 drive %2 fixed link voltage of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: An unequal fixed voltage MD1161: $MD_FIXED_LINK_VOLTAGE (DC link fixed voltage) has been found for axes of a drive module. Since a fixed voltage <> 0 replaces the measured DC link voltage value, but the DC link voltage value is measured only once for all axes of a drive module, the fixed voltage must be identical on all module axes before it is accepted.

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Set the same fixed voltage (MD 1161) on all module axes.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300888 Axis %1 drive %2 parameterization error in current control adaption

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The upper current limit in MD 1181: $MD_CURRCTRL_ADAPT_CURRENT_2 (upper adaption current limit) is less than the lower current limit in MD 1180: $MD_CURRCTRL_ADAPT_CURRENT_1 (lower adaption current limit).

Reactions: - Alarm display.
- Interface signals are set.

Remedy: Modify MD 1181: $MD_CURRCTRL_ADAPT_CURRENT_2 (upper adaption current limit) and MD 1180: $MD_CURRCTRL_ADAPT_CURRENT_1 (lower adaption current limit).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

© Siemens AG, 2002. All rights reserved

300900 Axis %1 drive %2 stop A triggered

Parameters: %1 = Axis number
%2 = Drive number

Definitions: The drive is disabled via STOP A. This blocks the pulses over the relay "Antrieb_IMP".
Request: In monitoring cycle.
If STOP A has been triggered, this can have several reasons:
1. The timer in MD 1356: $MD_SAFE_PULSE_DISABLE_DELAY of STOP B has expired.
2. The speed threshold in MD 1360: $MD_SAFE_STANDSTILL_VELO_TOL of STOP B has not been reached.
3. The test of the deactivation path has been requested by the user through SGE "Test stop selection", but the pulses were not deleted in MD 1357: $MD_SAFE_PULSE_DIS_CHECK_TIME when the time stage expired.
4. Safe brake ramp has responded.
5. "SG-specific stop reaction" is set to STOP A and has responded.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The user must find the cause and take appropriate measures.

Program Continuation: Switch control OFF - ON.

300901 Axis %1 drive %2 stop B triggered

Parameters: %1 = Axis number
%2 = Drive number

Definitions: The drive is disabled via STOP B. This blocks the pulses over the relay "Antrieb_IMP".
Request: In monitoring cycle.
If STOP B has been triggered, this can have several reasons:
1. Safe zero speed control has responded.
2. Call after STOP F, that means an error has occurred during cross-comparison.
3. "SG-specific stop reaction" is set to STOP B and has responded.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The user must check the cause and initiate the corresponding measures.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

300906 Axis %1 drive %2 safe braking ramp exceeded

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The drive is disabled via STOP A.
Request: In monitoring cycle.
The actual velocity of the axis has not been reduced during deceleration with "nset = 0" (STOP B or STOP C) but has increased over the follow-on velocity limit during deceleration and the tolerance entered in MD 1348: $MD_SAFE_VELO_TOL (actual velocity tolerance for SBR).
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the deceleration behavior and, if necessary, modify the velocity tolerance in MD 1348: $MD_SAFE_VELO_TOL. Restart is only possible with POWER ON.

Program Continuation: Switch control OFF - ON.

300907 Axis %1 drive %2 tolerance for safe operational stop exceeded

Parameters: %1 = Axis number
%2 = Drive number

Definitions: The drive is disabled via STOP A or STOP B. This blocks the pulses over the relay "Antrieb_IMP".
Request: In monitoring cycle.
The actual position has moved too far away from the setpoint/zero position (outside the zero speed window). The zero speed window is parameterized through MD 1330: $MD_SAFE_STANDSTILL_TOL (safe operational stop standstill tolerance).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the safe standstill tolerance: does the value match the precision and control dynamics of the axis? If not, increase the tolerance.

Program Continuation: Switch control OFF - ON.

300908 Axis %1 drive %2 stop C triggered

Parameters: %1 = Axis number
%2 = Drive number

Definitions: The drive is disabled via STOP C. At the end of the stop reaction, the drive remains under control, the axis is monitored for SBH.
Request: In monitoring cycle.

© Siemens AG, 2002. All rights reserved

If STOP C has been triggered, this can have several reasons (depending on the configuration):

1. The safe speed monitoring has been triggered (MD 1361: $MD_SAFE_VELO_STOP_MODE (SG-specific stop reaction) or MD 1363: $MD_SAFE_VELO_STOP_REACTION (SG-specific stop reaction) (840D as of SW4.2)).
2. The safe end-position monitoring has been triggered (MD 1362: $MD_SAFE_POS_STOP_MODE (safe end position stop reaction)).

The alarm indicates the initiation of a "deceleration at current limit" and the internal activation of "safe standstill".

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The user must check the cause and initiate the corresponding measures.

Program Continuation: Clear alarm with the RESET key. Restart part program

300909 Axis %1 drive %2 stop D triggered

Parameters: %1 = Axis number

%2 = Drive number

Definitions: The drive ws stopped by the NC with STOP D. At the end of the stop reaction, the drive remains under control, the axis is monitored for SBH.

Request: In monitoring cycle.

If STOP D has been triggered, this can have several reasons (depending on the configuration):

1. The safe speed monitoring has been triggered (MD 1361: $MD_SAFE_VELO_STOP_MODE (SG-specific stop reaction) or MD 1363: $MD_SAFE_VELO_STOP_REACTION (SG-specific stop reaction) (840D as of SW4.2)).
2. The safe end-position monitoring has been triggered (MD 1362: $MD_SAFE_POS_STOP_MODE (safe end position stop reaction)).

The alarm indicates the initiation of a "deceleration on path" in the NC and the internal activation of "safe standstill" in the NC and drive.

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The user must check the cause and initiate the corresponding measures.

Program Continuation: Clear alarm with the RESET key. Restart part program

300910 Axis %1 drive %2 stop E triggered

Parameters: %1 = Axis number

%2 = Drive number

Definitions: The drive ws stopped by the NC with STOP E. At the end of the stop reaction, the drive remains under control, the axis is monitored for SBH.

Request: In monitoring cycle.

If STOP E has been triggered, this can have several reasons (depending on the configuration):

1. The safe speed monitoring has been triggered (MD 1361: $MD_SAFE_VELO_STOP_MODE (SG-specific stop reaction).
2. The safe end-position monitoring has been triggered (MD 1362: $MD_SAFE_POS_STOP_MODE (safe end position stop reaction)).

© Siemens AG, 2002. All rights reserved

The alarm indicates the initiation of an "extended stop and retract" in the NC or "LIFT-FAST-ASUP" (840D) and the internal activation of "safe standstill" in the NC and drive.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. The user must check the cause and initiate the corresponding measures.

Program Continuation: Clear alarm with the RESET key. Restart part program

300911 Axis %1 drive %2 error in one monitoring channel

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The mutual comparison of the two monitoring channels has found a difference between input data or results of the monitoring operations. One of the monitors no longer functions reliably, i.e. safe operation is no longer possible.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department.

Find the difference between the monitoring channels. The error code indicating the cause is displayed as follows:

- On the 840D it is output in the alarm text.
- On the 840C MD 301: diagnostics for STOP F
- On the 611D MD 1395: $MD_SAFE_STOP_F_DIAGNOSIS (diagnostics for STOP F)

You can find the meaning of the error code as follows:

- On the 840D: description of alarm 27001
- On the 840C: description of alarms 1336* / 2097*

A possible cause is that the safety-related machine data are no longer identical or that the SGEs do not have the same level (recalibrate or check in the SI service display). If no error of this type is apparent, an error may have occurred in the CPU, e.g. a "flipped" memory cell. This error can be temporary (in this case it can be eliminated by a POWER ON) or permanent (if it occurs again after POWER ON replace the hardware).

Program Continuation: Clear alarm with the RESET key. Restart part program

300914 Axis %1 drive %2 safe velocity exceeded

Parameters: %1 = Axis number

%2 = Drive number

Definitions: The drive is disabled by the reaction configured in MD 1361: $MD_SAFE_VELO_STOP_MODE. At the end of the stop reaction, the drive remains under control, the axis is monitored for SBH.

Request: In monitoring cycle.

The axis has moved faster than allowed in machine data MD 1331: $MD_SAFE_VELO_LIMIT[n] (safe velocity limit values). If the "safe velocity correction" function is enabled in MD1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable), allowance must be made for the correction factor in the velocity limit when using SG2 and SG4.

Reactions:
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the input values of the machine data. Check the safe input signals: is the correct one of four velocity limits selected?

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the RESET key. Restart part program

300915 Axis %1 drive %2 safe end positions exceeded

Parameters: %1 = Axis number

%2 = Drive number

Definitions: The drive is disabled by the reaction configured in MD 1362: $MD_SAFE_POS_STOP_MODE. At the end of the stop reaction, the drive remains under control, the axis is monitored for SBH.

Request: In monitoring cycle.

The axis has exceeded the limit position which is entered in

- Modify MD 1334: $MD_SAFE_POS_LIMIT_PLUS[n] (upper limit for safe end position)
- Modify MD 1335: $MD_SAFE_POS_LIMIT_MINUS[n] (lower limit for safe end position)

Reactions: - NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. If no obvious operator error occurred: Check the input value of the machine data and check the SGEs: was the correct one of 2 end positions selected? If the MDs and SGEs are o.k., check the machine for any damage and rectify.

Program Continuation: Clear alarm with the RESET key. Restart part program

300950 Axis %1 drive %2 is not safely referenced

Parameters: %1 = Axis number

%2 = Drive number

Definitions: No stop reaction is initiated. The message will be present during enabling of the functions SN/SE until the axis status "Axis safely referenced" has been reached.

Request: In monitoring cycle.

1) The axis is not referenced or

2) The user enable for this axis is missing or was canceled. This can occur, for example, if the axis was moved after the machine was switched off and the standstill position which was stored is therefore no longer correct.

This message prompts the user to confirm the actual position. To do this, you must determine the position, e.g. as follows:

- Measure the position.
- Move to a known position.

Reactions: - Alarm display.

Remedy: Please inform the authorized personnel/service department. If no safe automatic referencing is possible, the user must confirm the new position via the soft key. This user confirmation marks this position as safe, that means the axis status "Axis safely referenced" is reached.

Warning:

If the axis has not been safely referenced and the user has not confirmed, the following applies:

- The safe cams are still active and not yet safe.
- The safe limit positions are not yet active.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300951 Axis %1 drive %2 test stop is running

Parameters: %1 = Axis number

%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The pulses are deleted.

1. If the positive acknowledgement is not received within the time configured in MD 1357: $MD_SAFE_PULSE_DIS_CHECK_TIME (time for checking the pulse deletion), STOP A is triggered.

2. If pulse deletion is acknowledged within the configured time in the drive, no stop reaction is triggered. This message will be displayed during selection via the SGE "Test stop selection" until the selection is canceled.

Request: In monitoring cycle.

The test stop has been activated by the user by setting the SGE "Test stop selection". The pulses are deleted.

1. If the positive acknowledgement is not received within the time configured in MD 1357: $MD_SAFE_PULSE_DIS_CHECK_TIME (time for checking the pulse deletion), STOP A is triggered.

2. If pulse deletion is acknowledged within the configured time in the drive, no stop reaction is triggered. This message will be displayed during selection via the SGE "Test stop selection" until the selection is canceled.

Reactions: - Alarm display.

Remedy: The message disappears automatically if the test is terminated by the user by clearing the SGE "Test stop selection". If STOP A is triggered, restart is only possible with POWER ON.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

300952 Axis %1 drive %2 acceptance test mode is active

Parameters: %1 = Axis number

%2 = Drive number

Definitions: The acceptance test mode has been activated by the user.

Reactions: - Alarm display.

Remedy: This message disappears automatically when the test is finished.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

301701 Axis %1 drive %2 limit value for safe velocity too large

Parameters: %1 = Axis number

%2 = Drive number

Definitions: Booting has been interrupted. The pulses remain disabled.

Request: In monitoring cycle.

The limit value of the safe velocity exceeds the velocity corresponding to a limit frequency of 200 kHz (300 kHz for 840D with SW4.2 and higher and for 840C with SW6.1 and higher).

The max. permissible speed that can be monitored is determined as follows:

nmax[rev/min] = (200000[Hz] * 60) / encoder marks value

Monitoring condition:

Modify MD 1331: $MD_SAFE_VELO_LIMIT[n] <= (1 / ue) * nmax

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check the entry in the machine data MD 1331: $MD_SAFE_VELO_LIMIT[n] (safe velocity limit values) correct, if necessary, and perform POWER ON.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

301702 Axis %1 drive %2 track inversion incorrect

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: With rotary spindles without EnDat interface operating with enhanced controller modules, the track inversion (MD_1011.0=1) must not be switched on. Otherwise this error is triggered.
Modify MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM)

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. With rotary spindles without EnDat interface operating with enhanced controller modules, the track inversion must be performed by soldering the A and B tracks differently: A <-> B and A* <-> B*

Program Continuation: Switch control OFF - ON.

301703 Axis %1 drive %2 encoder/motor type are not compatible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If a linear motor was selected (motor selection menu), but no linear scale was configured (MD_1011.4=0), or if a rotary motor was selected, but a linear scale configured (MD_1011.4=1), then this error is triggered.
Modify MD 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM)

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Encoder type must be parameterized according to the motor type.

Program Continuation: Switch control OFF - ON.

301704 Axis %1 drive %2 pole pair width/division of linear scale (internal) out of range

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: With linear motors the pole-pair width and graduations data is used for calculating the equivalent (internal) pole pair number and (internal) encoder marks. For this it is necessary that the encoder marks correspond to one or x pole pair widths as an integer. This error message is output if the result is not an integer pole pair width/graduations*x (to x=16) or if the calculated internal encoder marks value is too high. Absolute interpretation as an integer if the result is within a +/- 0.001 tolerance.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Please inform the authorized personnel/service department.

- Long traversing paths: It is advisable to use a length measuring system where the encoder marks match x* pole pair widths as integer.
- Short traversing paths: With short traversing paths, only a small error can accumulate and hardly affect the heating and the maximum power that can be reached if the encoder marks do not comply with the +/-0.001 tolerance. Then it is advisable to slightly modify the pole pair width:

Example:

Pole pair width: 56.8 mm, graduations: 2.7 µm

=> Pole pair number = 1, encoder marks = 21037.037 => error

Avoid the error by entering pole pair width = 56.7999 mm.

=> Pole pair number = 1, encoder marks = 21037.0 => no error

Program Continuation: Switch control OFF - ON.

301705 Axis %1 drive %2 distance-coded scale incorrectly parameterized

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: When selecting a distance-coded scale (MD_1011.7=1) it is also necessary to configure a length measuring system (MD_1011.4=1). In addition, the MDs 1040, 1041 and 1042 must not be zero or negative.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Check and, if necessary, correct MDs 1011: $MD_ACTUAL_VALUE_CONFIG (actual value sensing configuration IM), 1040, 1041 and 1042.

Program Continuation: Switch control OFF - ON.

301706 Axis %1 drive %2 parameterization of cam position invalid

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: At least one of the parameterized cams enabled via MD 1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable) does not comply with the rule stating that cam positions are not allowed to be within the tolerance range around the modulo position.

The valid tolerance range is:

- With inactive cam synchronization (MD 1301 bit 7 = 0):

Lower modulo value + POS_TOL <= cam position

Upper modulo value - POS_TOL > cam position

- With active cam synchronization (MD 1301 bit 7 = 1):

Lower modulo value + POS_TOL <= cam position

Upper modulo value - POS_TOL - CAM_TOL > cam position

Explanations:

- POS_TOL: Actual value tolerance (MD 1342: $MD_SAFE_POS_TOL (actual value tolerance cross-comparison))
- CAM_TOL: Cam tolerance (MD 1340: $MD_SAFE_CAM_TOL (tolerance for safe cams))
- Upper/lower modulo value: is determined by MD 1305: $MD_SAFE_MODULO_RANGE (actual value range for safe cam with rotary axes)

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department.
Check parameterization of the cam positions in
- Modify MD 1336: $MD_SAFE_CAM_POS_PLUS (plus cam position for safe cam) and
- Modify MD 1337: $MD_SAFE_CAM_POS_MINUS (minus cam position for safe cam) and perform POWER ON.
- Modify MD 1305: $MD_SAFE_MODULO_RANGE (actual value range for safe cam with rotary axes).

Program Continuation: Switch control OFF - ON.

301707 Axis %1 drive %2 parameterization of modulo value for safe cam (SN) invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The cam modulo range parameterized for a rotary axis via MD 1305: $MD_SAFE_MODULO_RANGE (actual value range for safe cam with rotary axes) violates the rule stating that only integral multiples of 360 degrees may be set.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Change parameterization of the cam modulo range in MD 1305: $MD_SAFE_MODULO_RANGE (actual value range for safe cam with rotary axes).

Program Continuation: Switch control OFF - ON.

301708 Axis %1 drive %2 actual value synchronization not allowed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The actual value synchronization for drift/slippage in MD 1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable) is deselected. This is only allowed with SBH/SG monitoring, since the absolute actual position is irrelevant for this type of monitoring. However, safe end position and/or cam monitoring is also selected.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Please inform the authorized personnel/service department. Deselect actual value synchronization for drift/slippage or the safe end position and/or safe cam monitoring in MD 1301: $MD_SAFE_FUNCTION_ENABLE (safe functions enable).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

301709 Axis %1 drive %2 submodule with integrated linearization invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If a submodule with integrated linearization is used, all submodules on the module must use the integrated linearization. A submodule with integrated linearization was found. However, not all submodules have this linearization.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Replace the submodule.

Program Continuation: Switch control OFF - ON.

301710 Axis %1 drive %2 resolution SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of the motor measuring system for an SSI encoder is incorrect: MD_1022 $MD_ENC_ABS_RESOL_MOTOR must not be 0.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Set MD_1022 $MD_ENC_ABS_RESOL_MOTOR to the correct value:
- Rotary encoder: Singleturn resolution (increments per revolution).
- Linear encoder: Resolution of an increment (in nanometers).

Program Continuation: Switch control OFF - ON.

301711 Axis %1 drive %2 transmission length SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of the motor measuring system for an SSI encoder is incorrect: MD_1028 $MD_NO_TRANSMISSION_BITS (SSI transmission length) is less than the number of all parameterized bits in MD_1021 $MD_ENC_ABS_TURNS_MOTOR (multiturn), MD_1022 $MD_ENC_ABS_RESOL_MOTOR (singleturn) and MD_1027 $MD_ENC_CONFIG bit 14 (alarm bit) and MD_1027 $MD_ENC_CONFIG bit 12 (parity bit).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Set parameters correctly for all associated machine data:
- MD_1028 $MD_NO_TRANSMISSION_BITS (SSI transmission length): number of bits in an SSI protocol, including all bits, such as alarm bit/parity bit
- MD_1021 $MD_ENC_ABS_TURNS_MOTOR (multiturn): number of resolvable revolutions

© Siemens AG, 2002. All rights reserved

- MD_1022 $MD_ENC_ABS_RESOL_MOTOR (singleturn): number of increments per revolution.
- MD_1027.Bit 12 $MD_ENC_CONFIG.Bit 12: parity bit
- MD_1027.Bit 14 $MD_ENC_CONFIG.Bit 14: alarm bit
- Example:
- SSI encoder with 25 bits transmission length, 12 bits multiturn, 12 bits singleturn, one alarm bit:
- $MD_NO_TRANSMISSION_BITS = 25
- $MD_ENC_ABS_TURNS_MOTOR = 4096
- $MD_ENC_ABS_RESOL_MOTOR = 4096
- $MD_ENC_CONFIG.Bit 14 = 1
- $MD_ENC_CONFIG.Bit 12 = 0

Program Continuation: Switch control OFF - ON.

301712 Axis %1 drive %2 multiturn SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of a linear SSI motor measuring system is incorrect: A linear measuring system cannot have any multiturn information.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Set MD_1021 $MD_ENC_ABS_TURNS_MOTOR (resolution number revolution) to 0.

Program Continuation: Switch control OFF - ON.

301713 Axis %1 drive %2 resolution SSI direct measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Configuration of the direct measuring system is faulty for SSI encoder: MD_1032 $MD_ENC_ABS_RESOL_DIRECT must not be 0.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Set MD_1032 $MD_ENC_ABS_RESOL_DIRECT to the correct value:
- Rotary encoder: Singleturn resolution (increments per revolution).
- Linear encoder: Resolution of an increment (in nanometers).

Program Continuation: Switch control OFF - ON.

301714 Axis %1 drive %2 transmission length SSI direct measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: Configuration of the direct measuring system is faulty for SSI encoder: MD_1041 $MD_NO_TRANSMISSION_BITS_DM (SSI transmission length) is smaller than the number of all parameterized bits in MD_1031 $MD_ENC_ABS_TURNS_DIRECT (multiturn), MD_1032 $MD_ENC_ABS_RESOL_DIRECT (singleturn) and MD_1037 $MD_ENC_CONFIG_DIRECT Bit 14 (alarm bit) and MD_1037 $MD_ENC_CONFIG_DIRECT Bit 12 (parity bit).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Set parameters correctly for all associated machine data:
- MD_1041 $MD_NO_TRANSMISSION_BITS_DM (SSI transmission length): number of bits in an SSI protocol, including all bits such as alarm bit and parity bit.
- MD_1031 $MD_ENC_ABS_TURNS_DIRECT (multiturn): number of resolvable revolutions
- MD_1032 $MD_ENC_ABS_RESOL_DIRECT (singleturn): number of increments per revolution
- MD_1037.Bit 12 $MD_ENC_CONFIG_DIRECT.Bit 12: parity bit
- MD_1037.Bit 14 $MD_ENC_CONFIG_DIRECT.Bit 14: alarm bit
- Example:
- SSI encoder with 25 bits transmission length, 12 bits multiturn, 12 bits singleturn, one alarm bit:
- $MD_NO_TRANSMISSION_BITS_DM = 25
- $MD_ENC_ABS_TURNS_DIRECT = 4096
- $MD_ENC_ABS_RESOL_DIRECT = 4096
- $MD_ENC_CONFIG_DIRECT.Bit 14 = 1
- $MD_ENC_CONFIG_DIRECT.Bit 12 = 0

Program Continuation: Switch control OFF - ON.

301715 Axis %1 drive %2 multiturn SSI direct measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Configuration faulty for a linear and direct measuring system with SSI: A linear measuring system cannot have any multiturn information.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Set MD_1031 $MD_ENC_ABS_TURNS_DIRECT (resolution number revolution) to 0.

Program Continuation: Switch control OFF - ON.

301716 Axis %1 drive %2 SSI direct measuring system without incremental signals not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: With the present module, it is not possible to use SSI encoders without incremental signals.

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Use newer module.

Program Continuation: Switch control OFF - ON.

301717 Axis %1 drive %2 SSI transmission timeout

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The SSI transmission must be completed within an NC clock cycle. This is not possible with its current parameterization.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Either increase the clock cycle length of the NC or increase the SSI transmission rate (MD_1030 $MD_ACTUAL_VALUE_CONFIG_DIRECT.Bits 14 and 15).
The following transmission rates are possible: 100 kHz, 500 kHz, 1 MHz and 2 MHz.
Notice: It might be possible that the length of the encoder cable does not allow an increase in frequency!

Program Continuation: Switch control OFF - ON.

301718 Axis %1 drive %2 combination of motor/power section invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The motor cannot be operated with the power section selected.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy:
- Check selection of power section.
- Check motor selection.
- Use valid power section.

Program Continuation: Switch control OFF - ON.

301719 Axis %1 drive %2 power section data incomplete

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The data for the power section are incomplete:
- Operation of a power section with an FDD motor requires: MD_1178 $MD_INVERTER_DERATING_SYN (derating factor at 8kHz)
- Operation of a power section with an MSD motor requires: MD_1179 $MD_INVERTER_DERATING_ASYN (derating factor at 8kHz)

© Siemens AG, 2002. All rights reserved

- Operation of a power section with a PE MSD motor requires: MD_1179 $MD_INVERTER_DERATING_ASYN (derating factor at 8kHz)
- MD_1175 $MD_INVERTER_THERM_CURR_ASYN (limit current power section for PE MSD)
- MD_1177 $MD_INVERTER_RATED_CURR_ASYN (limit current power section for PE MSD).

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Carry out new start-up with power section selection or enter the following data:
- Operation of a power section with an FDD motor requires:
- MD_1178 $MD_INVERTER_DERATING_SYN (derating factor at 8kHz)
- Operation of a power section with an MSD motor requires:
- MD_1179 $MD_INVERTER_DERATING_ASYN (derating factor at 8kHz)
- Operation of a power section with a PE MSD motor requires:
- MD_1179 $MD_INVERTER_DERATING_ASYN (derating factor at 8kHz)
- MD_1175 $MD_INVERTER_THERM_CURR_ASYN (limit current power section for PE MSD)
- MD_1177 $MD_INVERTER_RATED_CURR_ASYN (limit current power section for PE MSD).

Program Continuation: Switch control OFF - ON.

310505 Axis %1 drive %2 measuring circuit error of absolute track, code %3

Parameters:
%1 = NC axis number
%2 = Drive number
%3 = Fine error coding

Definitions:
- Absolute encoder (EQN 1325) Monitoring of the encoder hardware and the EnDat interface
- More accurate diagnostics via MD5023: $MD_ENC_ABS_DIAGNOSIS_MOTOR (diagnostics for absolute track on motor measuring system).
- Bit nos. and their meaning:
- Bit 0 Lighting failed
- Bit 1 Signal amplitude too small
- Bit 2 Code connection error
- Bit 3 Overvoltage
- Bit 4 Undervoltage
- Bit 5 Overcurrent
- Bit 6 Battery change necessary
- Bit 7 CRC error (evaluate bit 13), note: see below
- Bit 8 Encoder cannot be used, assignment of absolute track to incremental track not allowed
- Bit 9 C/D track for ERN 1387 encoder incorrect or EQN encoder connected
- Bit 10 Log cannot be aborted
- Bit 11 SSI level detected in data cable
- Bit 12 TIMEOUT while reading measured value
- Bit 13 CRC error (evaluate bit 7), note: see below
- Bit 14 (810D) Wrong IPU submodule for direct measuring system
- Bit 15 Encoder defective

© Siemens AG, 2002. All rights reserved

- CRC error: CRC error bit 7 and bit 13, meaning:
- Bit 7: 0, bit 13: 1 CRC error from SIDA-ASIC
- Bit 7:1, bit 13: 0 Control check byte error
- Bit 7: 1, bit 13: 1 Error on correction of absolute track by incremental track
- Bits 12 and 15: Zero level monitoring SSI
- Bits 14 and 15: Idle level monitoring SSI

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Check encoders, encoder lines and connectors between drive motor and 611D module. Check for temporary interruptions (loose contact) caused, for example, by movements in trailing cable. If necessary, replace the motor cable.
- Incorrect cable type
- Controller hardware not suitable for EnDat interface (e.g. control module with EPROM)

Program Continuation: Switch control OFF - ON.

310606 Axis %1 drive %2 external valve voltage supply failed

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The external 26.5 V supply is monitored for undervoltage in the closed-loop control.
Check the monitoring criteria:
- Voltage range (average) 26.0 V to 27.0 V
- Ripple factor 240 mVss
- No voltage failures

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The external 26.5 V supply is monitored for undervoltage in the closed-loop control.
Check the monitoring criteria:
- Voltage range (average) 26.0 V to 27.0 V
- Ripple factor 240 mVss
- No voltage failures

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310607 Axis %1 drive %2 valve not responding

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The valve is not responding to the valve slide setpoint. Cause: valve not connected or valve has no valve slide checkback signal.

© Siemens AG, 2002. All rights reserved

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy:
- Valve without valve slide checkback signal: Modify MD 5530: reset bit 2
- Check the valve connection.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310608 Axis %1 drive %2 speed controller at limit

Parameters:
%1 = NC axis number
%2 = Drive number

Definitions: The speed controller output is lying for an impermissibly long time at its limit (MD 5605: $MD_SPEEDCTRL_LIMIT_TIME and MD 1606: (speed controller limit threshold). The monitoring system is only active when the speed setpoint is below the speed threshold in MD 5606: SPEEDCTRL_LIMIT_THRESHOLD (speed controller limit threshold).

Reactions:
- The NC switches to follow-up mode.
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- Interface signals are set.
- Alarm display.
- NC Stop on alarm.
- Channel not ready.

Remedy:
- Is the drive blocked?
- Is the encoder connected? (check the encoder cable)
- Check the shield connection on the encoder cable
- Encoder defective?
- Check the encoder resolution
- The Uce monitoring circuit has been activated (perform a reset by switching the power supply off and on again).
- Replace the control module.
- Modify MD 5605: SPEEDCTRL_LIMIT_TIME and MD 5606: SPEEDCTRL_LIMIT_THRESHOLD in accordance with the mechanical and dynamic features of the axis.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310609 Axis %1 drive %2 encoder cut-off frequency exceeded

Parameters:
%1 = NC axis number
%2 = Drive number

Definitions: Actual velocity value exceeds encoder limit frequency fg,max = 650kHz

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy:
- The wrong encoder may be in use.
- Does the number of encoder lines match the setting in MD 5005: ENC_RESOL_MOTOR (encoder resolution for motor measuring system)?
- Is the encoder cable connected correctly?
- Is the encoder cable shield installed flat?
- Replace the encoder.
- Replace the 611D hydraulic module.
- Modify MD 5605: SPEEDCTRL_LIMIT_TIME and MD 5606: SPEEDCTRL_LIMIT_THRESHOLD in accordance with the mechanical and dynamic features of the axis.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310610 Axis %1 drive %2 wrong piston position

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The error is triggered if the actual position of the drive is negative.

Cause:
- Incorrect counting direction of actual position on drive side.
- Incorrect piston zero alignment.
- If the drive is referenced and the offset between the piston zero (piston stop at A side) and the machine zero is entered in MD 5040, the piston position in MD 5741 can only indicate positive values (from zero to piston stroke length).

Reactions:
- Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The counting direction for the actual position is correct on the drive side if:

1. Pos. setpoint voltage (e.g. function generator) -> cylinder piston travels from A to B. If not: Invert the actuation signal (change MD 5476 bit 0).

2. Cylinder piston travels from A to B -> v_act (MD 5707) > 0. If not: Invert the actual value (change MD 5011 bit 0).

Check the piston zero alignment and correct if necessary:

Set MD 5012 bit 14 and bit 15 to zero, save bootfile, reset NCK, perform a reference point approach and then align the position.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310611 Axis %1 drive %2 pressure sensor failed

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: Power limitation or friction compensation is activated: Modify MD 5241: bit 0 or bit 1 is set and both actual pressure values are less than 2% of the system pressure in MD 5101: WORKING_PRESSURE on performance enable.

Cause: Pressure sensor or connecting cable defective.

© Siemens AG, 2002. All rights reserved

Reactions: - Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Check the connection of both pressure sensors.
If no pressure sensors are installed:
• Deactivate the force limitation: Modify MD 5241: reset bit 0
• Deactivate friction compensation: Modify MD 5241: reset bit 1

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310612 Axis %1 drive %2 force limitation off

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The force limitation is deactivated.
Cause:
The force limitation is deactivated but:
• The NC has defined a force limit or
• Travel to fixed stop is selected.

Reactions: - Mode group not ready.
- Channel not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Activate power limitation: Modify MD 5241: Set bit 0.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. Restart part program.

310701 Axis %1 drive %2 speed controller cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: In the speed controller cycle drive MD 5001: SPEEDCTRL_CYCLE_TIME an impermissible value has been entered.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Permissible: 62.5µs <= T <= 500µs

Program Continuation: Switch control OFF - ON.

310702 Axis %1 drive %2 position controller cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

© Siemens AG, 2002. All rights reserved

Definitions: The monitor in the 611D module has detected a position controller pulse rate which is beyond the permissible limits.
The conditions for a permissible position controller pulse rate are:
1. Minimum cycle time: 250µs
2. Maximum pulse rate: 4 s
3. The position controller pulse rate must be a multiple of the speed controller cycle given in the drive MD 5001: SPEEDCTRL_CYCLE_TIME.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Change the position controller pulse rate on the NC.

Program Continuation: Switch control OFF - ON.

310703 Axis %1 drive %2 monitoring cycle time invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Monitoring cycle MD 5002: MONITOR_CYCLE_TIME (monitoring cycle) is invalid.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Please refer to the drive functions "FB/DB1" MD 1002.

Program Continuation: Switch control OFF - ON.

310704 Axis %1 drive %2 speed controller cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The speed controller cycle MD 5001: SPEEDCTRL_CYCLE_TIME must be identical for both axes on 2-axis modules.

Reactions:
- NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Set an identical speed controller cycle MD 5001: SPEEDCTRL_CYCLE_TIME for both axes.

Program Continuation: Switch control OFF - ON.

310705 Axis %1 drive %2 monitoring cycle times of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The monitoring cycle MD 5002: MONITOR_CYCLE_TIME must be identical for both axes on 2-axis modules.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Modify MD 5002: MONITOR_CYCLE_TIME for both axes.

Program Continuation: Switch control OFF - ON.

310706 Axis %1 drive %2 maximum working speed invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Because of the high maximum motor speed in the drive MD 5401: DRIVE_MAX_SPEED and the speed controller cycle in MD 5001: SPEEDCTRL_CYCLE_TIME sufficiently high speeds can occur to cause a format overflow.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Reduce the maximum working speed in MD 5401: DRIVE_MAX_SPEED or set a smaller speed controller cycle in MD 5001: SPEEDCTRL_CYCLE_TIME.

Program Continuation: Switch control OFF - ON.

310707 Axis %1 drive %2 STS configuration of axes differ

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of the control block MD 5003: STS_CONFIG (STS configuration) must be identical for both axes on 2-axis modules.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Check drive MD 5003: STS_CONFIG (STS configuration) and set the bits for the two axes of the module so that they are the same. (Do not change the default setting - this corresponds to the optimum configuration).

Program Continuation: Switch control OFF - ON.

310708 Axis %1 drive %2 number of encoder marks of measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The number of encoder marks of the motor measuring system in the drive MD 5005: ENC_RESOL_MOTOR (number of encoder marks of the motor measuring system) is zero or greater than the maximum input limit.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Match the number of encoder marks of the motor measuring system in drive MD 5005: ENC_RESOL_MOTOR (number of encoder marks of the motor measuring system) to the encoder in use. (Default setting for motor measuring system: (Default setting for motor measuring system: 2048 incr./rev.).

Program Continuation: Switch control OFF - ON.

310709 Axis %1 drive %2 error in piston diameter or piston rod diameter

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER is less than or equal to zero or the piston rod diameter in drive MD 5132: CYLINDER_PISTON_ROD_A_DIAMETER is greater than the piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER or the piston rod diameter in drive MD 5133: CYLINDER_PISTON_ROD_B_DIAMETER is greater than the piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER.

Reactions: - NC not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a valid piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER (0 < D <= 500mm). or Enter a piston rod diameter in drive MD 5132: CYLINDER_PISTON_ROD_A_DIAMETER which is less than the piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER or Enter a piston rod diameter in drive MD 5133: CYLINDER_PISTON_ROD_B_DIAMETER which is less than the piston diameter in drive MD 5131: CYLINDER_PISTON_DIAMETER.

Program Continuation: Switch control OFF - ON.

310710 Axis %1 drive %2 distance-coded scale incorrectly parameterized

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: If a distance-coded scale is selected (MD 5011 bit 7=1) a length measuring system must also be configured (MD 5011 bit 4=1).

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Check and, if necessary, correct MD 5011: ACTUAL_VALUE_CONFIG (actual value sensing configuration).

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

310750 Axis %1 drive %2 feedforward gain too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The feedforward control gain is calculated from the reciprocal of the gain in drive MD 5435: CONTROLLED_SYSTEM_GAIN.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Increase the speed controller cycle time in MD 5001: SPEEDCTRL_CYCLE_TIME.
Reduce the force controller feedforward factor in MD 5247: FORCE_FFW_WEIGHT.
Increase the gain in MD 5435: CONTROLED_SYSTEM_GAIN.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310751 Axis %1 drive %2 proportional gain for speed controller too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The P gain of the speed controller is too high:
- Modify MD 5406: SPEEDCTRL_GAIN_A (gain on A-side of cylinder edge)
- or MD 5407: SPEEDCTRL_GAIN (gain for piston adjustment with lowest natural frequency)
- or MD 5408: SPEEDCTRL_GAIN_B (gain on B-side of cylinder edge)

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a smaller value for the P gain of the speed controller:
- Modify MD 5406: SPEEDCTRL_GAIN_A (gain on A-side of cylinder edge)
- or MD 5407: SPEEDCTRL_GAIN (gain for piston adjustment with lowest natural frequency)
- or MD 5408: SPEEDCTRL_GAIN_B (gain on B-side of cylinder edge)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310752 Axis %1 drive %2 integral gain for speed controller invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The integral gain in MD 5409: SPEEDCTRL_INTEGRATOR_TIME cannot be represented.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Modify MD 5409: SPEEDCTRL_INTEGRATOR_TIME.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310753 Axis %1 drive %2 D component for speed controller invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The D component of the speed controller is too high:
- Modify MD 5431: SPEEDCTRL_DIFF_TIME_A (gain on A-side of cylinder edge)
- or MD 5432: SPEEDCTRL_DIFF_TIME (gain for piston adjustment with lowest natural frequency)
- or MD 5433: SPEEDCTRL_DIFF_TIME_B (gain on B-side of cylinder edge)

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a smaller value for the D component of the speed controller:
- Modify MD 5431: SPEEDCTRL_DIFF_TIME_A (gain on A-side of cylinder edge)
- or MD 5432: SPEEDCTRL_DIFF_TIME (gain for piston adjustment with lowest natural frequency)
- or MD 5433: SPEEDCTRL_DIFF_TIME_B (gain on B-side of cylinder edge)

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310754 Axis %1 drive %2 friction compensation gradient too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: Reduce the friction compensation gradient component MD 5460: FRICTION_COMP_GRADIENT is too high.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Reduce the friction compensation gradient component MD 5460: FRICTION_COMP_GRADIENT.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

310755 Axis %1 drive %2 area factor too high

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The positive area factor in drive MD 5462 AREA_FACTOR_POS_OUTPUT is too high or the negative area factor in drive MD 5463 AREA_FACTOR_NEG_OUTPUT is too high.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Select a smaller value for the positive area factor in drive MD 5462 AREA_FACTOR_POS_OUTPUT or select a smaller value for the negative area factor in drive MD 5463 AREA_FACTOR_NEG_OUTPUT.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310756 Axis %1 drive %2 controlled system gain is less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The controlled system gain in drive MD 5435: CONTROLLED_SYSTEM_GAIN is less than or equal to zero.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a valid controlled system gain in drive MD 5435: CONTROLLED_SYSTEM_GAIN (see model data calculations).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310757 Axis %1 drive %2 blocking frequency > Shannon frequency

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The blocking frequency of a speed setpoint filter or manipulated variable filter is greater than the Shannon sampling frequency from the sampling theorem.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy:
- The blocking frequency in drive MD 5514: SPEED_FILTER_1_SUPPR_FREQ
- or in drive MD 5210: OUTPUT_VCTRL_FIL_1_SUP_FREQ
- or in drive MD 5213: OUTPUT_VCTRL_FIL_2_SUP_FREQ
- or in drive MD 5268: FFW_FCTRL_FIL_1_SUP_FREQ
- or in drive MD 5288: OUTPUT_FIL_1_SUP_FREQ must be less than the reciprocal value of two speed controller cycle times MD 5001: SPEEDCTRL_CYCLE_TIME, i.e. less than 1 / (2 * MD 5001 * 31.25 microsec).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310758 Axis %1 drive %2 natural frequency > Shannon frequency

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The natural frequency of a speed setpoint filter is greater than the Shannon sampling frequency from the sampling theorem.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The natural frequency in Hz of a speed setpoint filter must be less than the reciprocal value of two speed controller cycles.

Speed filter:

MD 5520 * 0.01 * MD 5514 < 1 / (2 * MD 5001 * 31.25 microsec)
- BSP natural frequency drive MD 5520: SPEED_FILTER_1_BS_FREQ
- BSP blocking frequency drive MD 5514: SPEED_FILTER_1_SUPPR_FREQ
- Speed controller cycle drive MD 5001: SPEEDCTRL_CYCLE_TIME

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310759 Axis %1 drive %2 bandwidth numerator larger than double blocking frequency

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The bandwidth numerator of a speed or manipulated variable setpoint filter is greater than twice the blocking frequency.

This error message is only issued for the general bandstop filter if:
- Speed filter 1:
- MD 5516 > 0.0 or
- MD 5520 <> 100.0
- Manipulated variable filter 1:
- MD 5212 > 0.0
- Manipulated variable filter 2:
- MD 5215 > 0.0

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: The bandwidth numerator must be less than twice the blocking frequency.

Speed filter 1:

- BSP bandwidth numerator drive MD 5516: SPEED_FILTER_1_BW_NUMERATOR
- BSP blocking frequency drive MD 5514: SPEED_FILTER_1_SUPPR_FREQ, MD 5516 <= 2 * MD 5514

Manipulated variable filter 1:

- BSP bandwidth numerator drive MD 5212: OUTPUT_VCTRL_FIL_1_BW_NUM
- BSP blocking frequency drive MD 5210: OUTPUT_VCTRL_FIL_1_SUP_FREQ, MD 5212 <= 2 * MD 5210

Manipulated variable filter 2:

- BSP bandwidth numerator drive MD 5215: OUTPUT_VCTRL_FIL_2_BW_NUM
- BSP blocking frequency drive MD 5213: OUTPUT_VCTRL_FIL_2_SUP_FREQ, MD 5215 <= 2 * MD 5213

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310760 Axis %1 drive %2 bandwidth denominator greater than double natural frequency

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The bandwidth denominator of a speed or manipulated variable setpoint filter is greater than twice the natural frequency.

This error message is only issued for the general bandstop filter if:

- Speed filter 1:
- MD 5516 > 0.0 or
- MD 5520 <> 100.0

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The bandwidth denominator of a speed or manipulated variable setpoint filter must be less than twice the natural frequency.

- Speed filter 1:
- BSP bandwidth drive MD 5515: SPEED_FILTER_1_BANDWIDTH
- BSP blocking frequency drive MD 5514: SPEED_FILTER_1_SUPPR_FREQ
- BSP natural frequency drive MD 5520: SPEED_FILTER_1_BS_FREQ, MD5515 <= 2 * MD 5514 * 0.01 * MD 5520

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310761 Axis %1 drive %2 proportional gain of force controller too high

Parameters: %1 = NC axis number

%2 = Drive number

Definitions: The P gain of the force controller MD 5242: FORCECTRL_GAIN is too high.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a smaller value for the P gain of the force controller MD 5242: FORCECTRL_GAIN.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310762 Axis %1 drive %2 integral gain for force controller invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The integral gain in MD 5244: FORCECTRL_INTEGRATOR_TIME cannot be represented.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Modify MD 5244: FORCECTRL_INTEGRATOR_TIME.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310763 Axis %1 drive %2 D component of force controller invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The D component of the force controller MD 5246: FORCECTRL_DIFF_TIME is too high.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Enter a smaller value for the D component of the force controller MD 5246: FORCECTRL_DIFF_TIME.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310764 Axis %1 drive %2 controlled system gain for force controller is less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The controlled system gain for the force controller in drive MD 5240 FORCECONTROLLED_SYSTEM_GAIN is less than or equal to zero.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: Set a valid controlled system gain in drive MD 5240 FORCECONTROLLED_SYSTEM_GAIN (see model data calculations).

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310771 Axis %1 drive %2 gain in fine area of valve characteristic is less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The gradient in the fine area of the valve characteristic is less than or equal to zero.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The gradient in the fine area is calculated as follows:
- Positive quadrant: (MD 5464 - MD 5480) / (MD 5465 -5481)
- Negative quadrant: (MD 5467 - MD 5483) / (MD 5468 - 5484)

Enter a valid combination in the above drive MD.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310772 Axis %1 drive %2 gain in rough area of valve characteristic is less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The gradient in the coarse area of the valve characteristic is less than or equal to zero.

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The gradient in the coarse area is calculated as follows:
- Positive quadrant: (MD 5485 - MD 5464) / (MD 5486 -5465)
- Negative quadrant: (MD 5487 - MD 5467) / (MD 5488 - 5468)

Enter a valid combination in the above drive MD.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

© Siemens AG, 2002. All rights reserved

310773 Axis %1 drive %2 gain at end of saturation area of valve characteristic is less than or equal to zero

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The gradient at the end of the saturation area of the valve characteristic is less than or equal to zero. The saturation area is rounded by a parabola. The parabola has a maximum in the saturation area and can therefore not be inverted.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The gradient at the end of the saturation area is calculated as follows:
- Positive quadrant: 2 * (1.0 - MD 5485) / (1.0 - MD 5486) - (MD 5485 - MD 5464) / (MD 5486 -5465)
- Negative quadrant: 2 * (1.0 - MD 5487) / (1.0 - MD 5488) - (MD 5487 - MD 5467) / (MD 5488 - 5468)

Enter a valid combination in the above drive MD.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310774 Axis %1 drive %2 zero area and knee area of valve characteristic overlap

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The zero area and the knee area of the valve characteristic overlap.

Reactions:
- NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The zero area and the knee area overlap if:
- Positive quadrant: (MD 5481 + MD 5482) > (MD 5465 - 5466)
- Negative quadrant: (MD 5484 + MD 5482) > (MD 5468 - 5466)

Enter a valid combination in the above drive MD.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

310775 Axis %1 drive %2 knee area and saturation area of valve characteristic overlap

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The knee area and the saturation area of the valve characteristic overlap.

© Siemens AG, 2002. All rights reserved

Reactions: - NC not ready.
- Channel not ready.
- Channel not ready.
- NC Stop on alarm.
- NC Start disable in this channel.
- The NC switches to follow-up mode.
- Alarm display.
- Interface signals are set.

Remedy: The knee area and the saturation area overlap if:
- Positive quadrant: (MD 5465 + MD 5466) > MD 5486
- Negative quadrant: (MD 5468 + MD 5466) > MD 5488

Enter a valid combination in the above drive MD.

Program Continuation: Clear alarm with the RESET key in all channels. Restart part program.

311710 Axis %1 drive %2 resolution SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of the motor measuring system for an SSI encoder is incorrect: MD_5022 $MD_ENC_ABS_RESOL_MOTOR must not be 0.

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Set MD_5022 $MD_ENC_ABS_RESOL_MOTOR to correct value:
- Rotary encoder: Singleturn resolution (increments per revolution).
- Linear encoder: Resolution of an increment (in nanometers).

Program Continuation: Switch control OFF - ON.

311711 Axis %1 drive %2 transmission length SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of the motor measuring system for an SSI encoder is incorrect: MD_5028 $MD_NO_TRANSMISSION_BITS (SSI frame length) is smaller than the number of all parameterized bits in MD_5021 $MD_ENC_ABS_TURNS_MOTOR (multiturn), MD_5022 $MD_ENC_ABS_RESOL_MOTOR (singleturn) and MD_5027 $MD_ENC_CONFIG bit 14 (alarm bit) and MD_5027 $MD_ENC_CONFIG bit 12 (parity bit).

Reactions: - Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Set parameters correctly for all associated machine data:
- MD_5028 $MD_NO_TRANSMISSION_BITS (SSI frame length): number of bits in an SSI protocol, including all bits, such as alarm bit/parity bit
- MD_5021 $MD_ENC_ABS_TURNS_MOTOR (multiturn): no. of resolvable revolutions
- MD_5022 $MD_ENC_ABS_RESOL_MOTOR (singleturn): number of increments per revolution
- MD_5027.Bit 12 $MD_ENC_CONFIG.Bit 12: parity bit
- MD_5027.Bit 14 $MD_ENC_CONFIG.Bit 14: alarm bit

© Siemens AG, 2002. All rights reserved

Example:
SSI encoder with 25 bits message frame length, 12 bits multiturn, 12 bits singleturn, one alarm bit:
- $MD_NO_TRANSMISSION_BITS = 25
- $MD_ENC_ABS_TURNS_MOTOR = 4096
- $MD_ENC_ABS_RESOL_MOTOR = 4096
- $MD_ENC_CONFIG.Bit 14 = 1
- $MD_ENC_CONFIG.Bit 12 = 0

Program Continuation: Switch control OFF - ON.

311712 Axis %1 drive %2 multiturn SSI motor measuring system invalid

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: The configuration of a linear SSI motor measuring system is incorrect: A linear measuring system cannot have any multiturn information.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Set MD_5021 $MD_ENC_ABS_TURN_MOTOR (number of representable revolutions) to 0.

Program Continuation: Switch control OFF - ON.

311716 Axis %1 drive %2 SSI measuring system without incremental signals not possible

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: With the present module, it is not possible to use SSI encoders without incremental signals.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

Remedy: Use newer module.

Program Continuation: Switch control OFF - ON.

311717 Axis %1 drive %2 SSI transmission timeout

Parameters: %1 = NC axis number
%2 = Drive number

Definitions: SSI transmission must be able to finish within one position control cycle. This is not possible with its current parameterization.

Reactions:
- Mode group not ready.
- Channel not ready.
- NC Start disable in this channel.
- NC Stop on alarm.
- Alarm display.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Remedy: Either increase the position control cycle of the NC or increase the SSI transmission rate (MD_5011 $MD_ACTUAL_VALUE_CONFIG bits 14 and 15).
The following transmission rates are possible: 100 kHz, 500 kHz, 1 MHz and 2 MHz.
Notice: It might be possible that the length of the encoder cable does not allow an increase in frequency!

Program Continuation: Switch control OFF - ON.

380001

Profibus-DP: startup error, reason %1 parameter %2 %3 %4.

Parameters: %1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Definitions: An error occurred during startup of the DP master.
Overview: Cause of the error, Par 1, Par 2, Par 3:

- 01 = DPM version, DPM version, DPA version, --
- 02 = DPM boot-up timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM boot-up status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM boot-up error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = DPM-PLL sync error, --, --, --
- 07 = alarm queue too long, Actual number, Setpoint number, --
- 08 = unknown client, Client ID, --, --
- 09 = Client version, Client ID, Client version, DPA version
- 10 = too many clients, Client number, Max. number of clients, --

Clients are the following components of the control system that use the PROFIBUS DP:
Client ID = 1: PLC
Client ID = 2: NCK
The following can be causes:

- Error in contents of SDB1000
- Corruption of parts of system program
- Hardware defect on NC component

Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: Follow the following steps:
1. Check the control project (particularly SDB1000); check MD 11240; if a user-specific SDB1000 is in use, load it again.
2. If the error still exists, save the data and restart the control system with the default values with which the system was supplied.
3. If the system starts up without an error, the user data should be loaded one step at a time.
4. If the error still exists after startup with the default values, reboot from the PC card or update the software.
5. If the error still exists, replace the hardware.
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

© Siemens AG, 2002. All rights reserved

380003 Profibus-DP: operating error, reason %1, parameter %2 %3 %4.

Parameters:
%1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Definitions:
An operating error occurred on the PROFIBUS DP in cyclic mode.
Overview: Cause of the error, Par 1, Par 2, Par 3:
- 01 = unknown alarm, Alarm class, Logical address, --
- 02 = DPM cycle timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM cycle status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM cycle error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = client not registered, Client number, Max. number of clients, --
- 06 = synchronization error, Number of sync violation, --, --

Alarm class: (see alarm 380060)
The following can be primary causes:
- For error cause 01: Data transfer error on PROFIBUS DP
- For error cause 02, 03, 04: Error in contents of SDB1000
- For error cause 02, 03, 04, 05: Corruption of parts of system program
- For error cause 06: The PCI bus cycle does not match the expected rate and synchronization is not possible for this reason. The correct PCI bus cycle must be entered.

The error can also be caused by a hardware problem on the MCI module.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy:
- For error cause 01:
- Check the electrical and fault-related specifications for PROFIBUS DP, assess the cable installation
- Check the terminating resistors of the PROFIBUS connectors (ON setting at end of cable, otherwise OFF setting required)
- Check slave
- For error cause 02, 03, 04:
- Check SDB1000
- For error cause 02, 03, 04, 05:
- Follow the procedure described for troubleshooting alarm 380 001
- For error cause 06:
- The correct PCI bus cycle must be entered.

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation:
Clear alarm with the RESET key. Restart part program

380005 Profibus-DP: bus access conflict, type %1, counter %2

Parameters:
%1 = Conflict type
%2 = Serial number within a conflict sequence

Definitions:
An access conflict occurred on the PROFIBUS DP in cyclic mode: The NCK attempted to write data to the bus or to read from the bus while cyclic data transfer was active. This can lead to data integrity problems.
Type 1: Cyclic transfer has not finished on the PROFIBUS when the NCK attempts to read data.

© Siemens AG, 2002. All rights reserved

Type 2: The NCK has not finished writing data when cyclic transfer begins again. Counter %2 contains a serial number starting at 1. A maximum of 10 alarms are output in succession. If no conflicts occur in a DP cycle, the counter is reset and new alarms are output again on the next conflict.

Reactions: - Alarm display.

Remedy:
- Check the timing again, in particular ensure that the settings in SYSCLOCK_CYCLE_TIME and POSCTRL_CYCLE_DELAY are correct: POSCTRL_CYCLE_DELAY must be larger for type 1. POSCTRL_CYCLE_DELAY must be smaller for type 2.
- If alarm-free operation cannot be achieved with any POSCTRL_CYCLE_DELAY setting, SYSCLOCK_CYCLE_TIME must be increased.
- If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Clear alarm with the Delete key or NC START.

380020 Profibus-DP: SDB1000 error %1 for SDB source %2

Parameters: %1 = Cause of the error

%2 = SDB1000 source

Definitions: Error in SDB1000 for PROFIBUS DP configuration.

Error cause:
- 01 = SDB1000 does not exist in SDB1000 source.
- 02 = SDB1000 in SDB1000 source too large.
- 03 = SDB1000 in SDB1000 source cannot be activated.

SDB1000 source:
- 00 = Default SDB (selected by MD 11240 = 0 if no user SDB1000 is loaded on the control system)
- 01 = Standard SDB1 (selected by MD 11240 = 1)
- 02 = Standard SDB2 (selected by MD 11240 = 2)
- .....
- 100 = SDB1000 stored in battery-backed memory (SRAM)
- 101 = User SDB1000 stored in file system
- 102 = SDB1000 reloaded in SRAM on startup

Reaction: PROFIBUS DP is inactive or operating in accordance with default SDB1000.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy:
- Check MD 11240.
- If SDB1000 source = 100: Reload user SDB1000 in passive file system /_N_IBN_DIR/_N_SDB1000_BIN.
- If SDB1000 source = 101: Check backup batteries.
- If SDB1000 source = 102: Follow the procedure described for troubleshooting alarm 380 001.
- If alarm 380 021 is also signaled, please follow the instructions provided for this alarm.

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380021 Profibus-DP: default SDB1000 was loaded

Definitions: No user-specific SDB1000 exists. The default SDB1000 was loaded during startup. Without process peripherals, the NC is ready for a start-up. The alarm is triggered the first time the NC is switched on or once if the SDB1000 stored in the supported RAM is lost.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.

Remedy: Create the user-specific SDB1000 and load it on the control system, or select and activate it via MD 11240 standard SDB1000. Restart the NC. If the error occurs the next time the NC is switched on, the SDB1000 which was loaded contains an error and must be created again.

Program Continuation: Clear alarm with the Delete key or NC START.

380022 Profibus-DP: configuration of DP master has been changed

Definitions: The PROFIBUS configuration on the DP master was changed during operation, e.g. by downloading a new hardware configuration via Step7. Since it is possible that the cycle data have changed, operation cannot be continued and a warm start is required.

If the DP master functionality is within the PLC (as on the 840Di), the PLC will have been stopped for the download and alarm 2000 (PLC sign-of-life) output.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: NCK restart

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380040 Profibus-DP: configuration error %1, parameter %2

Parameters: %1 = Cause of the error

%2 = Parameter

Definitions: The PROFIBUS DP was not generated in SDB1000 in accordance with the configuration specifications of the NC in use.

Overview: Cause of the error, Par 1:

- 01 = SDB1000 contains slave without diagnosis slot, Slave address
- 02 = SDB1000 contains too many slot entries, Identifier
- 03 = SDB1000 contains no equidistance data, No fct.

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: Check that SDB1000

- contains a diagnostic slot for every slave and
- contains only slave entries relevant to the application.

It is possible to include a superset of slaves in SDB1000 for use in different end versions of the product. This overloads the NC memory and runtime capacity, however, and should therefore be avoided.

If this alarm occurs, it is necessary to reduce SDB1000 to a minimum.

If the code for the error cause is 03, check that equidistance is activated in the SDB (using Step7 HW Config).

If the alarm continues to occur, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380050 Profibus-DP: multiple assignment of inputs on address %1

Parameters: %1 = Logical address

© Siemens AG, 2002. All rights reserved

Definitions: Multiple assignments of input data have been detected in the logical address space. Logical address: Base address of the address area defined several times

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: The address partitioning should be checked as follows:

Check for multiple assignments in the following machine data:

- MD 13050[1] - MD 13050[n]: n = highest axis index on control system
- MD 12970, 12971: PLC address area for digital inputs
- MD 12978, 12979: PLC address area for analog inputs

If no inconsistencies can be found in the parameters, compare the machine data with the configuration in SDB1000. In particular, check that the lengths configured for the individual slots do not result in area overlaps. When you find the cause of the error, change the machine data and/or SDB1000.

Program Continuation: Switch control OFF - ON.

380051 Profibus-DP: multiple assignment of outputs on address %1

Parameters: %1 = Logical address

Definitions: Multiple assignments of input data have been detected in the logical address space. Logical address: Base address of the address area defined several times

Reactions:
- Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: The address partitioning should be checked as follows:

Check for multiple assignments in the following machine data:

- MD 13050[1] - MD 13050[n]: n = highest axis index on control system
- MD 12974, 12975: PLC address area for digital outputs
- MD 12982, 12983: PLC address area for analog outputs

If no inconsistencies can be found in the parameters, compare the machine data with the configuration in SDB1000. In particular, check that the lengths configured for the individual slots do not result in area overlaps. When you find the cause of the error, change the machine data and/or SDB1000.

Program Continuation: Switch control OFF - ON.

380060 Profibus-DP: alarm %1 on logical address %2 from unassigned station

Parameters: %1 = Alarm class

%2 = Logical address

Definitions: SDB1000 contains a slave which is not assigned in the NC via the MD parameters (see the help for alarm 380 050/51). The slave is also connected to the PROFIBUS DP. An alarm has been triggered by a slave of this type.

Alarm class:

- 01 = Station return (or arrival)
- 02 = Station failure

Operation with the NC is not possible.

Reactions: - Alarm display.

Remedy:
- Enter machine data or
- Modify SDB1000 or
- Disconnect the slave from PROFIBUS DP or
- Acknowledge the alarm.

© Siemens AG, 2002. All rights reserved

Program Continuation: Clear alarm with the Delete key or NC START.

380070 Profibus DP: no input slot available for base address %1 (length %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
Length=1 indicates a digital input.
Length=2 indicates a analog input.

Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTIN.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTIN.
- NCK restart

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380071 Profibus DP: no output slot available for base address %1 (size %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
For length =1 it is a digital output,
For length =2 it is an analog output.

Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380072 Profibus DP: output slot for base address %1 (size %2) not allowed

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was set for a digital or analog output, the area is resides in the access range of the PLC (PIQ, base addresses < 128).
For length =1 it is a digital output,
For length =2 it is an analog output.

© Siemens AG, 2002. All rights reserved

Reactions: - Alarm display.
- Interface signals are set.
- Channel not ready.
- NC Start disable in this channel.

Remedy: Only use addresses >= 128 for output slots.
Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart

If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380075 Profibus DP: DP I/O failure slave %1

Parameters: %1 = Slave address

Definitions: Failure of a PROFIBUS slot used by the NCK for digital or analog I/O.

Reactions: - Alarm display.

Remedy: Check that the PROFIBUS slave is operating correctly (all slaves must be included in the bus, green LED).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

380500 Profibus-DP: fault on drive %1, code %2, value %3, time %4

Parameters: %1 = Axis
%2 = Fault code of drive (P824)
%3 = Fault value of drive (P826)
%4 = Fault time of drive (P825)

Definitions: Contents of fault memory of assigned drive.

Reactions: - Alarm display.

Remedy: See drive documentation for fault codes/fault values.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action

400102 Delete DB 2 in the PLC and restart

Definitions: -

Reactions: - -

Remedy: See the machine manufacturer's information.

Program Continuation: Internal

400103 Delete DB 3 in the PLC and restart

Definitions: -

Reactions: - -

Remedy: See the machine manufacturer's information.

Program Continuation: Internal

400106 Delete DB 3 in the PLC and restart

Definitions: -

Reactions: - -

Remedy: See the machine manufacturer's information.

Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

400109 Delete DB 9 in the PLC and restart

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400171 Delete DB 71 in the PLC and restart

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400172 Delete DB 72 in the PLC and restart

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400173 Delete DB 73 in the PLC and restart

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400174 Delete DB 74 in the PLC and restart

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400250 NCK sign-of-life monitoring

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400251 NCK has not started up

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400252 Sign-of-life monitoring

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.

© Siemens AG, 2002. All rights reserved

Program Continuation: Internal

400260 Failure of machine control panel 1

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400261 Failure of machine control panel 2

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400262 Failure of manual operating device

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400604 Set change with M06 in the machine data

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

400902 Impermissible channel no. parameter in FC 9

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401502 Impermissible axis no. parameter in FC 15

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401602 Impermissible axis no. parameter in FC 16

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401702 Impermissible spindle IF no. parameter in FC 17

Definitions: -

© Siemens AG, 2002. All rights reserved

Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401805 Impermissible axis no. parameter in FC 18

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401901 Impermissible BAG no. parameter in FC 19

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

401902 Impermissible channel no. parameter in FC 19

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

402501 Impermissible BAG no. parameter in FC 25

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

402502 Impermissible channel no. parameter in FC 25

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

410150 Area in M group decoder list is too large

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810001 Error OB_event

Definitions: -
-
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

810002 Synchronous error

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810003 Asychronous error

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810004 Stop/Interrupt event

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810005 Order form execution event

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810006 Error communication event

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810007 Error H/F system event

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810008 Error diagnostics data from modules

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

810009 User-diagnostics event

Definitions: -
Reactions: - -
Remedy: See the machine manufacturer's information.
Program Continuation: Internal

© Siemens AG, 2002. All rights reserved

1.3 List of action numbers

The following list describes the actions stated in the alarm texts under "Action %.." according to their numbers.

No. 1

Explanation	Run Init phase (tasks are initialized after Power On)
Not allowed if	-
Remedy	-

No. 2

Explanation	Perform Reset (VDI signal: Reset, mode group Reset or after Power On)
Not allowed if	-
Remedy	-

No. 3

Explanation	Activate Reset Init blocks (VDI signal: Reset)
Not allowed if	-
Remedy	-

No. 4

Explanation	Perform Reset, end of program has been detected (NC block with M30)
Not allowed if	-
Remedy	-

No. 5

Explanation	Change the mode to the MDA or AUTOMATIC program operating mode (VDI signal: mode group)
Not allowed if	1. The channel is active (program running, block search, loading machine data) 2. The other program operating mode has already been started. 3. A channel has exited the mode group due to an interrupt. 4. Overstore or digitizing has been selected.
Remedy	> Abort the program (Reset key) or stop the program (not with block search, loading MD) > Abort the program (Reset key) > Abort the program with the Reset key or wait until the interrupt is terminated. > Deactivate overstore, digitizing

No. 6

Explanation	Automatic change from an internal mode to the mode that was externally set (with TEACH_IN an attempt is made after every stop to change from the internal mode "AUTOMATIC, MDA" to TEACH_IN)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 7

Explanation	Change the mode to a manual mode (VDI signal (mode group): JOG, TEACH_IN, REF)
Not allowed if	1. Nesting depth too great: The current processing operation can be interrupted by various events (e.g. interrupt). Depending on the event, asynchronous subroutines are activated. These asynchronous subroutines can be interrupted in the same manner as the user program. Unlimited nesting depth is not possible for asynchronous subroutines due to memory limitations. Example: An interrupt interrupts the current program processing. Further higher-priority interrupts will interrupt the asynchronous programs which have been activated. 2. The channel is active (program running, block search, loading machine data) 3. A channel has exited the mode group due to an interrupt. 4. Overstore or digitizing has been selected.
Remedy	> Abort the program with the Reset key > Abort the program with the Reset key or stop the program (not with block search, loading MD) > Abort the program with the Reset key or wait until the interrupt is terminated. > Deactivate overstore, digitizing

No. 8

Explanation	Activate overstore (PI command).
Not allowed if	-
Remedy	-

No. 9

Explanation	Activate overstore (PI command).
Not allowed if	-
Remedy	-

No. 10

Explanation	Perform user interrupt "Asynchronous subroutine" (VDI signal: digital-analog interface, asynchronous subroutine interface)
Not allowed if	1. The channel is active due to block search or loading machine data 2. The channel is stopped and the asynchronous subroutine "ASUP_START_MASK" must be started and the current block cannot be reorganized. 3. Digitizing has been selected. 4. Reference point approach has not been performed yet. 5. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Wait until the block search or loading MD is completed, or abort program (Reset key) > Activate a block change until the NC block can be reorganized. > Deactivate digitizing > Perform reference point approach or ignore this state via the MD "ASUP_START_MASK". > Abort program

No. 11

Explanation	Perform user interrupt "ASUP" with rapid retraction (VDI signal: digital-analog interface)
Not allowed if	See 10
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 12

Explanation	Perform user interrupt at block end (VDI signal: ASUP interface, digital-analog interface)
Not allowed if	See 10
Remedy	-

No. 13

Explanation	Perform rapid retraction (VDI signal: digital-analog interface and asynchronous subroutine interface, for further actions see 10, 11, 12, 85, 86)).
Not allowed if	-
Remedy	-

No. 14

Explanation	Move tool - only with tool management (PI command)
Not allowed if	-
Remedy	-

No. 15

Explanation	Perform delete distance-to-go or axis synchronization (VDI signal: delete distance-to-go or follow-up mode) (follow-up mode: e.g. when switching on an axis movement)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Abort program > Abort program

No. 16

Explanation	Abort the subroutine repetition (VDI signal: delete subroutine number of passes)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Abort program > Abort program

No. 17

Explanation	Abort subroutine execution. (VDI signal: abort program level)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Abort program > Abort program

No. 18

Explanation	Activate single block (VDI signal: activate single block)
Not allowed if	-
Remedy	-

No. 19

Explanation	Deactivation of single block. (VDI signal: activate single block)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 20

Explanation	Activate main run single block. (OPI variable and VDI signal: activate single block)
Not allowed if	-
Remedy	-

No. 21

Explanation	Activate decoding single block. (OPI variable and VDI signal: activate single block)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program > Abort program

No. 22

Explanation	Activate main program single block. (OPI variable and VDI signal: activate single block)
Not allowed if	-
Remedy	-

No. 23

Explanation	Activate traversing single block. (OPI variable and VDI signal: activate single block)
Not allowed if	-
Remedy	-

No. 24

Explanation	Start program processing. (VDI signal: NC Start)
Not allowed if	1. Program status active 2. An alarm reaction is pending which prevents a start or forces braking. 3. Reference point approach has not been performed yet.
Remedy	- > Execute condition for clearing alarm > Reference point approach

No. 25

Explanation	Start program processing. (Channel communication, NC block: Start)
Not allowed if	1. Program status active 2. An alarm reaction is pending which prevents a start or forces braking. 3. Reference point approach has not been performed yet. 4. An incorrect operating mode has been selected (AUTOMATIC only).
Remedy	> Protect Start with WAITE > Execute condition for clearing alarm > Reference point approach > Select program operating mode

No. 26

Explanation	Start continuation of program processing. (VDI signal: NC Start)
Not allowed if	1. Program status active 2. An alarm reaction is pending which prevents a start or forces braking. 3. Reference point approach has not been performed yet.
Remedy	- > Execute condition for clearing alarm > Reference point approach

© Siemens AG, 2002. All rights reserved

No. 27

Explanation	Start continuation of the selected processing -JOG, reference point or digitizing. (VDI signal: NC Start)
Not allowed if	1. Jog motion active 2. An alarm reaction is pending which prevents a start or forces braking.
Remedy	- > Execute condition for clearing alarm

No. 28

Explanation	Start processing in the digitizing submode. (VDI signal: NC Start)
Not allowed if	1. Jog motion active 2. An alarm reaction is pending which prevents a start or forces braking. 3. Reference point approach has not been performed yet.
Remedy	- > Execute condition for clearing alarm > Reference point approach

No. 29

Explanation	Stop all axes. (VDI signal: Stop All or via Reset key)
Not allowed if	-
Remedy	-

No. 30

Explanation	Perform a program stop. (NC block: M0)
Not allowed if	-
Remedy	-

No. 31

Explanation	Stop the JOG motion. (VDI signal: NC Stop)
Not allowed if	-
Remedy	-

No. 32

Explanation	Stop digitizing processing. (VDI signal: NC Stop)
Not allowed if	-
Remedy	-

No. 33

Explanation	Start the selected processing. (VDI signal: NC Start)
Not allowed if	1. Process change active (operating mode change, activate/deactivate digitizing/overstore) 2. An alarm reaction is pending which prevents a start or forces braking. 3. A process is running (NC program, block search, loading machine data)
Remedy	- > Execute condition for clearing alarm -

No. 34

Explanation	Stop the active processing. (VDI signal: NC Stop)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 35
Explanation: Start machine data processing (INI file is already in the NCK). (PI command)
Not allowed if: -
Remedy: -

No. 36
Explanation: Start machine data processing (INI file is external, e.g.: on MMC) (PI command)
Not allowed if: -
Remedy: -

No. 37
Explanation: Stop because of mode group single block. VDI signal, single type A (only executable blocks), after stop in another channel in this mode group
Not allowed if: -
Remedy: -

No. 38
Explanation: Stop because of mode group single block. VDI signal, single type B (any blocks), after stop at block end in another channel in this mode group
Not allowed if: -
Remedy: -

No. 39
Explanation: Stop because end of overstore buffer "_N_OSTOREXX_SYF" has been reached.
Not allowed if: -
Remedy: -

No. 40
Explanation: Start preprocessing (NC block: Stopre)
Not allowed if: -
Remedy: -

No. 41
Explanation: Stop processing at block end. (NC block: M00/M01)
Not allowed if: -
Remedy: -

No. 42
Explanation: Stop processing at block end. (Alarm, VDI signal: NC stop at block end)
Not allowed if: -
Remedy: -

No. 43
Explanation: Stop at end of asynchronous subroutine, if start was performed from "stopped".
Not allowed if: -
Remedy: -

No. 44
Explanation: Activate program. (PI command)
Not allowed if: -
Remedy: -

© Siemens AG, 2002. All rights reserved

No. 44
Explanation Activate program. (PI command)
Not allowed if -
Remedy -

No. 45
Explanation Activate the program which is still external. (PI command)
Not allowed if -
Remedy -

No. 46
Explanation Program selection from another channel. (Channel communication, NC block: INIT)
Not allowed if -
Remedy -

No. 47
Explanation Save definition of an asynchronous subroutine which can be activated. (PI command)
Not allowed if -
Remedy -

No. 48
Explanation Sets all machine data with the attribute (NEW_CONF) active. (PI command)
Not allowed if -
Remedy -

No. 49
Explanation Clear all alarms with the clear condition CANCELCLEAR (PI command, acknowledge alarm key)
Not allowed if -
Remedy -

No. 50
Explanation Continue block search. (NC block, Stopre)
Not allowed if -
Remedy -

No. 51
Explanation Start block search. (PI command)
Not allowed if -
Remedy -

No. 52
Explanation Continue block search. (PI command)
Not allowed if -
Remedy -

No. 53
Explanation Activate digitizing. (PI command)
Not allowed if -
Remedy -

© Siemens AG, 2002. All rights reserved

No. 54

Explanation	Deactivate digitizing. (PI command)
Not allowed if	-
Remedy	-

No. 55

Explanation	Switch on function generator. (PI command)
Not allowed if	-
Remedy	-

No. 56

Explanation	Switch off function generator. (PI command)
Not allowed if	-
Remedy	-

No. 57

Explanation	Wait for a program marker. (Channel communication, NC block: WAITM)
Not allowed if	-
Remedy	-

No. 58

Explanation	Wait for end of program. (Channel communication, NC block: WAITE)
Not allowed if	-
Remedy	-

No. 59

Explanation	Program selection from another channel, synchronous. (Channel communication, NC block: INIT + SYNC)
Not allowed if	-
Remedy	-

No. 60

Explanation	Wait until receive acknowledge from MMC. (NC block, MMC_CMD)
Not allowed if	-
Remedy	-

No. 61

Explanation	Activate the skip block function (VDI signal: skip block)
Not allowed if	Nesting depth too great
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program

No. 62

Explanation	Deactivate the skip block function (VDI signal: skip block)
Not allowed if	Nesting depth too great
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program

© Siemens AG, 2002. All rights reserved

No. 63

Explanation	Activate test run. (VDI signal: rapid traverse override)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program > Abort program

No. 64

Explanation	Deactivate test run (VDI signal: rapid traverse override)
Not allowed if	1. Nesting depth too great 2. The active block, after which deceleration takes place, cannot be reorganized (occurs when deceleration takes place over several blocks).
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program > Abort program

No. 65

Explanation	Activate read-in disable for main run block. (VDI signal: read-in disable)
Not allowed if	-
Remedy	-

No. 66

Explanation	Deactivate read-in disable for main run block. (VDI signal: read-in disable)
Not allowed if	-
Remedy	-

No. 67

Explanation	Stop at block end. (Alarm)
Not allowed if	-
Remedy	-

No. 68

Explanation	Stop all axes. (Alarm)
Not allowed if	-
Remedy	-

No. 69

Explanation	Activate program test. (VDI signal: program test)
Not allowed if	1. Tool management is active. 2. The NCK channel state is not ready
Remedy	> Save tool data > Abort the program or process with the Reset key or wait for end of program.

No. 70

Explanation	Deactivate program test. (VDI signal: Program Test)
Not allowed if	The NCK channel state is not ready
Remedy	> Abort the program or process with the Reset key or wait for end of program.

No. 71

Explanation	Stop at the end of block preparation. (Alarm)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 72

Explanation	Stop at the end of block preparation with subsequent reorganization of block processing. (Alarm)
Not allowed if	Nesting depth too great
Remedy	> Wait until the preceding asynchronous subroutine is terminated or abort the program

No. 73

Explanation	Conditional stop at block end. (If, after continuation by means of an NC start, there is still a reason to stop "Stop at block end", the program stops again).
Not allowed if	-
Remedy	-

No. 74

Explanation	Conditional stop at block end. (Despite the start, the interpreter or the preprocessing does not manage to put a block in main run)
Not allowed if	-
Remedy	-

No. 75

Explanation	Stop preprocessing. (Alarm)
Not allowed if	-
Remedy	-

No. 76

Explanation	Retraction with G33 and Stop.
Not allowed if	-
Remedy	-

No. 77

Explanation	Conditional wait for program marker (NC block: WAITMC)
Not allowed if	-
Remedy	-

No. 78

Explanation	Set marker. (NC block: SETM)
Not allowed if	-
Remedy	-

No. 79

Explanation	Clear marker (NC_block: CLEARM)
Not allowed if	-
Remedy	-

No. 80

Explanation	Selection of an NC block. (PI command)
Not allowed if	-
Remedy	-

No. 81

Explanation	Disable the NC program which is currently being processed for editing. (PI command)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 82

Explanation	Start a program in the TEACH IN submode. (VDI-Signal: NC-Start
Not allowed if	See 33 and 5
Remedy	-

No. 83

Explanation	Start a program in the TEACH IN submode. (VDI signal: NC Start)
Not allowed if	See 33 and 5
Remedy	-

No. 84

Explanation	Reorganize block processing.
Not allowed if	-
Remedy	-

No. 85

Explanation	Activate user interrupt "asynchronous subroutine" in a manual mode. (VDI signal: asynchronous subroutine interface, digital-analog interface)
Not allowed if	See 10
Remedy	-

No. 86

Explanation	Activate user interrupt "Asynchronous subroutine" Is only executed with READY channel status (VDI signal: asynchronous subroutine interface, digital-analog interface)
Not allowed if	See 10
Remedy	-

No. 87

Explanation	Perform user interrupt "Asynchronous subroutine". (VDI signal: digital-analog interface and asynchronous subroutine interface, for further actions see 10, 11, 12, 85, 86).
Not allowed if	-
Remedy	-

No. 88

Explanation	Stop processing. (VDI signal: mode group Stop)
Not allowed if	-
Remedy	-

No. 89

Explanation	Set all machine data with the attribute (NEW_CONF) active. (NC block: NEW_CONF)
Not allowed if	-
Remedy	-

No. 90

Explanation	Set all machine data with the attribute (NEW_CONF) active. (NC block: NEW_CONF at block search)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 91

Explanation	Start continuation of interpreter processing. (internal preprocessing stop)
Not allowed if	-
Remedy	-

No. 92

Explanation	Save interlock for data
Not allowed if	The channel is not in the "stopped" state
Remedy	-

No. 93

Explanation	Activate user data, e.g. via MMC; newly modified tool lengths are effective immediately in the running program.
Not allowed if	1. The channel is not in the "stopped" state 2. The channel is stopped and the current block cannot be reorganized.
Remedy	> Press the Stop/Single-Block/ Reset/StopAtEnd (Automatic) key > Activate a block change until the NC block can be reorganized.

No. 94

Explanation	Write the user PLC version into the version file
Not allowed if	-
Remedy	-

No. 95

Explanation	Switch measuring systems (PI command)
Not allowed if	-
Remedy	-

No. 96

Explanation	Shut down system (VDI signal)
Not allowed if	-
Remedy	-

No. 97

Explanation	Activate block search program invocation in mode 5. This mode simulates the block search in which the program under "Program test operation" is processed as far as the target of the block search.
Not allowed if	-
Remedy	-

No. 98

Explanation	Extended stop and retract
Not allowed if	-
Remedy	-

No. 99

Explanation	Block search (general) is currently being activated (negative acknowledgement may be output for PI service).
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

No. 100

Explanation	Integrated block search, i.e. a block search is restarted on a stopped program.
Not allowed if	-
Remedy	-

No. 101

Explanation	External zero offset is activated via the PLC. Movement is stopped, a Reorg is performed, the interpreter is switched over and then selected using REPOS and continued automatically.
Not allowed if	1. The channel is not in AUTO or MDA. 2. The channel is stopped and the current block cannot be reorganized.
Remedy	> Select Auto or MDA. > Activate a block change until the NC block can be reorganized.

No. 102

Explanation	Single block type 3 is activated. With single block type 3, a stop is performed at all main blocks. Unlike single block type 1, the part program command SBLOF is ignored.
Not allowed if	-
Remedy	-

No. 103

Explanation	Stopping of a single axis movement (VDI signal)
Not allowed if	The axis is not controlled by the PLC (exception: previous response for oscillation axis).
Remedy	-

No. 104

Explanation	Stopping of a single axis movement by an alarm
Not allowed if	The axis is not controlled by the PLC (exception: previous response for oscillation axis).
Remedy	-

No. 105

Explanation	Continuation of a single axis movement (VDI signal)
Not allowed if	The axis has not been stopped previously. Not for all axis types at present.
Remedy	-

No. 106

Explanation	Interruption of a single axis movement (VDI signal)
Not allowed if	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 107

Explanation	Deletion of distance-to-go of a single axis movement (VDI signal)
Not allowed if	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 108

Explanation	Activation: Axis is now controlled by the PLC (VDI signal)
Not allowed if	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy:	-

© Siemens AG, 2002. All rights reserved

No. 109

Explanation	Power Off: Axis is now controlled by the PLC (VDI signal)
Not allowed if	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 115

Explanation	The event if triggered by the positive PLC edge of the signal "Repos-Mode-Edge".
Not allowed if	The channel is active (program running, block search, loading machine data)
Remedy	> Abort the program with the Reset key or stop the program (not with block search, loading machine data)

No. 116

Explanation	Activate tool management commands. (Ch. VDI signal)
Not allowed if	The NCK channel state is not ready
Remedy	> Abort the program or process with the Reset key or wait for end of program.

No. 117

Explanation	Deactivate tool management commands. (Ch. VDI signal)
Not allowed if	The NCK channel state is not ready
Remedy	> Abort the program or process with the Reset key or wait for end of program.

No. 118

Explanation	Selection of desired safety limitations (SGE) (always allowed)
Not allowed if	-
Remedy	-

© Siemens AG, 2002. All rights reserved

1.4 Error codes of alarm 300500

After the error codes 1 listed below an error code 2 may be specified. This must be interpreted as the hexadecimal number of the terminal block/module.

Error code	**0001H**
Extension 1	-
Extension 2	Incorrect address
Explanation	When the program memory was being tested during power-up, it was detected that the written bitmap could not be read back. Cause: Hardware error on control module. Error only occurs with drive software V1.x (EPROM version). The error has been replaced by system errors F034 and F035 in subsequent download versions.
Remedy	Replace the control module.

Error code	**0002H**
Extension 1	-
Extension 2	Incorrect address
Explanation	When the data memory was being tested during power-up, it was detected that the written bitmap could not be read back. Cause: Hardware error on control module.
Remedy	Replace the control module.

Error code	**0003H**
Extension 1	1 = Bit 0 : Level 3 time slice (MONCYC) (MD 1300) 2 = Bit 1 : 4 msec time slice 4 = Bit 2 : 1 msec time slice 8 = Bit 3 : PC time slice 10 = Bit 4 : SC time slice (MD 1001) 20 = Bit 5 : IC time slice (MD 1000) 40 = Bit 6 : SI time slice (MD 1300) A0 = Startup, synchronization B0 = Background computing time
Extension 2	-
Explanation	The computing time of the drive processor is no longer sufficient for the selected functions in the specified cycle times. This error normally only occurs in conjunction with start-up functions in the case of default values (FFT measurement, step response)... SINUMERIK Safety Integrated: Monitoring cycle too short
Remedy	During startup with FFT or measurement of step response: - Deactivate emergency retraction (MD 1636) - Switch off feedforward control (MD 1004.0) - Deactivate MIN-MAX memory (MD 1650.0) - Reduce the number of DAC output channels (max. 1 channel) - Deactivate variable signaling function (MD 1620.0) - Deactivate encoder phase error compensation (MD 1011.1) - Increase the position control cycle time on the NC - Increase the time slice during which the system error occurred or - Increase lower-level time slices - Deselect functions which are no longer required - Use performance instead of standard control module

© Siemens AG, 2002. All rights reserved

Error code	**0004H**
Extension 1	-
Extension 2	-
Explanation	Upon servo enable, the NC must update the sign-of-life monitoring in each position control cycle. In case of error, sign-of-life monitoring has not been updated for at least two consecutive position control cycles. Causes: NC failure, communication failure on the drive bus. Hardware error on drive module or hardware error on NC CPU if error occurs sporadically at intervals of several hours. Other cause: Ring programming for GI (gear interpolation) and gantry axes.
Remedy	Check cable connections, perform remedial measures (check shielding or ground connection). Replace the NC hardware, replace the control module. Replace the NC CPU with the "VB" version, replace the control module.

Error code	**0005H**
Extension 1	1A : DS <> 1 if CS = 0 2A : DS <> 1, 2, 3, 4, 5 2B : DS-CS <> 0, 1 2C : DS = 3 on PO parameterization error 3A : DS <> 1, 2, 3, 4, 5 3B : DS-CS <> 0, 1
Extension 2	-
Explanation	The power-up of the drive modules is grouped into 5 states (steps). The states are specified consecutively by the NC and acknowledged by the drive. An error occurs when an invalid specified state is detected in the drive. Causes: Communication failure on the drive bus. Hardware error on the drive module, hardware error on the NC.
Remedy	Check cable connections, perform remedial measures (check shielding or ground connection). Replace the control module, replace the NC hardware.

Error code	**0006H**
Extension 1	-
Extension 2	-
Explanation	The infinite loop for processing communication has been exited. The cause is probably a hardware error on the control module.
Remedy	Replace the control module.

Error code	**0007H**
Extension 1	-
Extension 2	-
Explanation	An illegal state was read by the hardware during the cycle synchronization of the NC and the drive. Synchronization could not be performed.
Remedy	Replace the control module.

© Siemens AG, 2002. All rights reserved

Error code	**0010H**
Extension 1	1 = Hardware underflow 2 = Hardware overflow 3 = Software underflow 4 = Software overflow
Extension 2	-
Explanation	The limits of the internal processor hardware stack or of software stack in the data memory have been violated. The cause is probably a hardware error on the control module.
Remedy	Reload drive software. Replace the control module.

Error code	**0011H**
Extension 1	Opcode address
Extension 2	-
Explanation	The watchdog on the control module has timed out. The cause is a hardware error in the time base on the control module.
Remedy	Replace the control module.

Error code	**0012H**
Extension 1	-
Extension 2	-
Explanation	The NC basic cycle which is generated on the NC and transferred to the drive via the drive bus cable has failed. Possible causes: NCK reset, EMC interference, NC hardware error, cable break on the drive bus, hardware error on the control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection). Replace the NC hardware, replace the control module.

Error code	**0013H**
Extension 1	-
Extension 2	-
Explanation	The NC basic cycle which is generated on the NC and transferred to the drive via the drive bus cable has supplied a pulse which does not match the clock timing. Possible causes: EMC interference on drive bus, NC hardware error, hardware error on the control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection). Replace the NC hardware, replace the control module.

Error code	**0014H**
Extension 1	Incorrect address
Extension 2	-
Explanation	The processor has detected an illegal command in the program memory.
Remedy	Replace the control module.

© Siemens AG, 2002. All rights reserved

Error code	**0015H**
Extension 1	-
Extension 2	Version 4.0 and higher: segment of the code/data range containing the error, whereby: 0: P: memory 1: X: memory 2: Y: memory
Explanation	A difference has been detected between the setpoint and actual checksums during continuous testing of the checksum. The cause is probably a hardware error on the control module.
Remedy	Replace the control module.

Error code	**0016H**
Extension 1	-
Extension 2	-
Explanation	An illegal processor interrupt has occurred. The cause is probably a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections. Replace the control module.

Error code	**0017H**
Extension 1	-
Extension 2	-
Explanation	A disallowed processor interrupt has occurred. The cause is probably a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections. Replace the control module.

Error code	**0018H**
Extension 1	-
Extension 2	-
Explanation	An illegal processor interrupt has occurred. The cause is probably a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections. Replace the control module.

Error code	**0019H**
Extension 1	-
Extension 2	-
Explanation	An illegal processor interrupt has occurred. The cause is probably a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections. Replace the control module.

© Siemens AG, 2002. All rights reserved

Error code	**001BH**
Extension 1	0: Deviation from current 0 1: Module selection does not match the existing hardware (V 2.6 and higher)
Extension 2	NC drive number
Explanation	When the actual current measuring is starting up and during cyclical operation with a pulse disable, current 0 is expected as the system ensures that no currents can flow. Deviation from current 0: The hardware for actual current measurement may be defective. Module selection does not match the existing hardware: If a single-axis power section has been addressed as a two-axis power section by the module selection (software parameterization of power section), this system error is output via actual current measurement because a current > 0 is measured.
Remedy	Deviation from current 0: Replace the control module. Check the cable connections. Module selection does not match the existing hardware: - Change software parameterization of power section (two-axis power section -> single-axis power section) - 2. Define axis as inactive or use a two-axis power section.

Error code	**0020H**
Extension 1	-
Extension 2	-
Explanation	The NC has attempted to activate the second axis on a single-axis module. Possible communication failure on the drive bus or control module defective.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0021H**
Extension 1	-
Extension 2	-
Explanation	The NC has attempted to activate two axes on a single-axis module. Possible communication failure on the drive bus or control module defective.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0022H**
Extension 1	-
Extension 2	-
Explanation	The motor measuring system is either not fitted or is defective for at least one axis of the drive module. As the configuration of the measuring systems is detected by the NC and communicated to the drive, communication failures on the drive bus can also cause this error.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

© Siemens AG, 2002. All rights reserved

Error code	**0023H**
Extension 1	Read K1C register of the relevant PCU ASIC
Extension 2	NC drive number
Explanation	The motor measuring system has a motor encoder with a voltage output. A corresponding IPU submodule with a voltage input is required. An unexpected submodule has been detected.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0024H**
Extension 1	-
Extension 2	-
Explanation	An illegal internal axis number was detected during software processing. (Only 0 or 1 is permitted in the case of 2-axis modules). Possible causes: Defective control module, EMC interference
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0025H**
Extension 1	-
Extension 2	-
Explanation	An illegal internal physical axis number was detected during software processing. Possible causes: Defective control module, EMC interference
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0026H**
Extension 1	-
Extension 2	NC drive number
Explanation	The NC is attempting to register an FDD module as an MSD. Possible communication failures on the drive bus or control module defective.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0027H**
Extension 1	-
Extension 2	NC drive number
Explanation	The NC is attempting to register an MSD module as an FDD. Possible communication failures on the drive bus or control module defective.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0028H**
Extension 1	Read K1C register of the relevant PCU ASIC
Extension 2	NC drive number
Explanation	Only certain submodules are permitted for the direct measurement system. An illegal submodule has been detected.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

© Siemens AG, 2002. All rights reserved

Error code	**0030H**
Extension 1	0x01 ;Unsupported ROSCTR 0x02 ;Illegal ROSCTR 0x03 ;Job management "defective" 0x04 ;Incorrect PDUREF on acknowledgement 0x05 ;Acknowledgement not permitted at this time 0x06 ;Acknowledgement is not supported 0x07 ;Illegal PROTID 0x08 ;Illegal PARLG (uneven) 0x09 ;Buffer management "defective" 0x0A ;Illegal PI identifier (internal) 0x0B ;Illegal internal state of PI restart 0x0C ;Sequential circuit in WRITEDATA "defective" 0x0D ;Illegal parameter on REFRESH_PIZUST
Extension 2	NC drive number
Explanation	Either irrecoverable communications errors were detected on the drive bus or the drive software is no longer consistent. The cause is either a faulty drive bus interface or a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection). Replace the control module.

Error code	**0031H**
Extension 1	0x40 ;Illegal PDU length 0x41 ;Axes do not have the same PDU length 0x42 ;PDU length is not a word multiple 0x43 ;Axes do not have the same NC type
Extension 2	-
Explanation	The NC has transferred illegal key data for communication via the drive bus. The cause is probably either faults on the drive bus or a defective control module.
Remedy	Replace the control module. Check cable connections, perform remedial measures (check shielding or ground connection).

Error code	**0032H**
Extension 1	0x20 ;Job management "defective" 0x21 ;Illegal state in RESET_TRANSPO 0x22 ;Checksum incorrect more than 3 times 0x23 ;Receive PDU too long 0x24 ;State 6XX-Abort illegal
Extension 2	NC drive number
Explanation	Either irrecoverable communications errors were detected on the drive bus or the drive software is no longer consistent. The cause is either a faulty drive bus interface or a hardware error on the control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection). Replace the control module.

© Siemens AG, 2002. All rights reserved

Error code	**0033H**
Extension 1	0x51 ;Incorrect data format in element list 0x52 ;Incorrect conversion group specified in Refresh
Extension 2	-
Explanation	The drive software is no longer consistent. The cause is probably a hardware error on the control module.
Remedy	Reload drive software. Replace the control module.

Error code	**0034H**
Extension 1	0 or incorrect address
Extension 2	0x60 ;Illegal SERVO response on STF handshake 0x61 ;Error during RAM check 0x62 ;Transport checksum does not correspond to that of the SERVO.
Explanation	Errors were detected on loading the drive software. The cause is either faults during the transfer via the drive bus or a defective control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection), replace control module.

Error code	**0035H**
Extension 1	0 or incorrect address
Extension 2	0x60 ;Illegal SERVO response on STF handshake 0x61 ;Error during RAM check 0x62 ;Transport checksum does not correspond to that of the SERVO.
Explanation	Errors were detected on loading the drive software. The cause is either faults during the transfer via the drive bus or a defective control module.
Remedy	Check the drive bus cable and cable connections, perform remedial measures (check shielding or ground connection), replace control module.

Error code	**0040H**
Extension 1	-
Extension 2	-
Explanation	An illegal number of current setpoint filters (> 4) has been entered.
Remedy	Correct number of current setpoint filters (MD 1200)

Error code	**0041H**
Extension 1	-
Extension 2	-
Explanation	An illegal number of speed setpoint filters (> 2) has been entered.
Remedy	Correct the number of speed setpoint filters (MD 1500)

© Siemens AG, 2002. All rights reserved

Error code	**0044H**
Extension 1	-
Extension 2	NC drive number
Explanation	Rotor position synchronization contains an error (drive software 2.5 only). The difference between the first part of rotor position synchronization (coarse synchronization) and the second part (fine synchronization to the active encoder zero marker) is greater than 45 electrical. An excessive difference can occur due to: - Incorrect encoder alignment - EMC problems on the zero marker signal - Voltage level of C/D track too high
Remedy	- Check encoder alignment and EMC measures - New startup - Check MODE - Replace motor

Error code	**0045H**
Extension 1	-
Extension 2	NC drive number
Explanation	The NC has entered either an encoder with distance-coded reference point markers or a BERO proximity switch in register $1D of the motor measuring system of the PCU ASIC. This is not allowed during fine synchronization which is activated by power-up, by zero monitoring errors or by deselection of the parking axis.
Remedy	The NC/PLC is not allowed to enter an encoder with distance-coded reference point markers or a BERO proximity switch in register $1D of the motor measuring system of the PCU ASIC following power-up, zero monitoring errors or deselection of the parking axis.

Error code	**0046H**
Extension 1	-
Extension 2	-
Explanation	With drive software 4.02 and higher a drive power-up is only possible when the drive software is being loaded.
Remedy	Reload drive software.

© Siemens AG, 2002. All rights reserved

1.5 System reactions on alarms

Identifier COMPBLOCKWITHREORG
Effect Block preparation has detected an error which can be rectified by a program modification. Reorganization is performed after a program modification.
- Correction block with reorganization.

Identifier COMPENSATIONBLOCK
Effect Block preparation has detected an error which can be rectified by a program modification.
- Correction block

Identifier FOLLOWUP
Effect Follow-up of axes
- NC switches to follow-up mode

Identifier INTERPRETERSTOPP
Effect Program execution is aborted after all the prepared blocks (interpolator buffer) have been processed.
- Interpreter stop

Identifier LOCALREACTION
Effect - Local alarm reaction

Identifier NOALARMREACTION
Effect - No alarm reaction

Identifier NOREADY | NCKREACTIONVIEW
Effect NCK ready off: Active rapid deceleration (i.e. with maximum braking current) of all drives Clearing of servo enable for all NC axes Release of NC ready relay
- NC not ready

Identifier NOREADY | BAGREACTIONVIEW
Effect Mode group ready off: Active rapid deceleration (i.e. with maximum braking current) of the drives in this mode group Clearing of servo enable for the NC axes concerned.
- Mode group not ready

Identifier NOREADY
Effect Channel ready off: Active rapid deceleration (i.e. with maximum braking current) of the drives in this channel Clearing of servo enable for the NC axes concerned.
- Channel not ready

Identifier NONCSTART
Effect It is not possible to start a program in this channel.
- NC Start disable in this channel.

Identifier NOREFMARK
Effect The axes in this channel have to be referenced.
- Reference axes in this channel.

Identifier SETVDI
Effect VDI interface signal alarm is set.
- Interface signals are set.

© Siemens AG, 2002. All rights reserved

Identifier SHOWALARM
Effect Alarm is displayed on MMC.
- Alarm display

Identifier STOPBYALARM
Effect Ramp stop of all channel axes.
- NC Stop on alarm

Identifier STOPATENDBYALARM
Effect Stop at end of block.
- NC Stop on alarm at end of block

Identifier SHOWALARMAUTO
Effect The alarm is displayed whenever bit 0 of machine data ENABLE_ALARM_MASK is set. The reaction should be set whenever an alarm should only occur during automatic mode without manual operation by the user.
- Alarm reaction in automatic mode

Identifier SHOWWARNING
Effect The alarm is displayed whenever bit 1 of machine data ENABLE_ALARM_MASK is set. It is designed for warnings which should normally be suppressed.
- Alarm display

Identifier ALLBAGS_NOREADY
Effect The Ready is canceled in all mode groups. The reaction thus corresponds to an NCKREACTIONVIEW|NOREADY, the difference being that the NC READY relay is not canceled and the corresponding VDI bit is not set. This is desirable in the event of an emergency stop for example.
- Mode group not ready

Identifier DELAY_ALARM_REACTION
Effect If this alarm reaction is configured in the alarm handler, all alarm reactions by alarms which now occur are buffered channel-specifically and are therefore not active. The alarms are displayed on the MMC. Mode group and NC-wide reactions are transferred. The reaction is cleared by activating the call clearDelayReaction or by an alarm which has configured NO_DELAY_ALARM_REACTION. This activates all the delayed alarm reactions.
- All channel-specific alarm reactions delayed on alarm, alarm display

Identifier NO_DELAY_ALARM_REACTION
Effect The DELAY_ALARM_REACTION state is canceled.
- The alarm reaction delay is canceled.

Identifier ONE_IPO_CLOCK_DELAY_ALARM_REACTION
Effect All alarm reactions are delayed by one cycle when an alarm is output. This functionality became necessary as part of ESR development.
- All alarm reactions are delayed by one IPO cycle on alarm.

© Siemens AG, 2002. All rights reserved

1.6 Cancel criteria for alarms

Identifier CANCELCLEAR

Effect The alarm is cleared by pressing the Cancel key in any channel. It is also cleared by the Start part program key.
- Clear the alarm with the Clear key or with NC START.

Identifier CLEARHIMSELF

Effect Self-clearing alarm. The alarm is cleared not by an operator action but explicitly by a "clearAlarm" in programmed the NCK source code.
- Alarm display showing cause of alarm disappears. No further operator action necessary.

Identifier NCSTARTCLEAR

Effect The alarm is cleared by starting a program in the channel in which the alarm occurred. The alarm is also cleared by an NC reset.
- Clear the alarm with NC START or the RESET key and continue the program.

Identifier POWERONCLEAR

Effect The alarm is cleared by switching the control off/on.
- Switch control OFF - ON.

Identifier RESETCLEAR

Effect The alarm is cleared by pressing the Reset key in the channel in which the alarm occurred.
- Clear alarm with the RESET key. Restart part program.

Identifier BAGRESETCLEAR

Effect The alarm is cleared by a "BAGRESETCLEAR" command or by carrying out a reset in all channels of this mode group.
- Press the RESET key to clear alarm in all channels of this mode group. Restart part program.

Identifier NCKRESETCLEAR

Effect The alarm is cleared by an "NCKRESETCLEAR" command or by carrying out a reset in all channels.
- Clear alarm in all channels with the RESET key. Restart part program.

Identifier NOCLEAR

Effect The clear information is only required for the internal pseudo alarm number EXBSAL_NOMOREALARMS.

© Siemens AG, 2002. All rights reserved

Appendix 2

A Abbreviations

ASCII	American Standard Code for Information Interchange
AV	Job planning
BA	Mode
BAG	Mode groups
BB	Ready
BCD	Binary Coded Decimals
CNC	Computerized Numerical Control
CP	Communication Processor
CPU	Central Processing Unit
CR	Carriage Return
CRC	Cutter Radius Compensation
CSB	Central Service Board: PLC module
CTS	Clear To Send: Signal from serial data interfaces
DAC	Digital-to-Analog Converter
DB	Data Block
DIN	German standard

© Siemens AG, 2002. All rights reserved

DIO	Data Input/Output: Data transfer display
DRF	Differential Resolver Function: handwheel displacement
DRY	Dry Run
DSB	Decoding Single Block
DSR	Data Send Ready: Signal from serial data interfaces
DW	Data Word
EIA Code	Special punched tape code, number of holes per character always odd
EPROM	Erasable Programmable Read Only Memory
ETC	ETC key: Soft key bar extension in the same menu
FDB	Product database
FIFO	First in First Out: Memory that works without addresses and whose data is read in the same sequence as stored.
FM	Function Module
FM-NC	Function Module Numerical Control
FRA	Frame block
FRAME	Coordinate conversion with the components zero (work) offset, rotation, scaling, mirroring
FSD	Feed Spindle Drive
FST	Feed Stop
GUD	Global User Data
GWRC	Grinding Wheel Radius Compensation

© Siemens AG, 2002. All rights reserved

HD	Hard Disk
HHU	Handheld Unit
HMS	High-resolution Measuring System
HW	Hardware
I	Input
IM	Interface Module
IM S/R	Interface Module (Send/Receive)
INC	Incremental Mode
I/RF	Infeed/Regenerative Feedback unit
ISO code	Special punched tape code, number of holes per character always even
K_{UE}	Transmission ratio
K_V	Servo gain factor
K1...K4	Channel 1 to channel 4
LAD	LADder diagram
LCD	Liquid Crystal Display: Opto-electronic display with liquid crystals
LEC	Leadscrew Error Compensation
LED	Light Emitting Diode: Display
LUD	Local User Data
MB	Megabyte
MC	Measuring Circuit

© Siemens AG, 2002. All rights reserved

MCP	Machine Control Panel
MD	Machine data
MDA	Manual Data Automatic
MLFB	Order No. (machine readable)
MMC	Man-Machine Communication: Operator interface in numerical control for operation, programming and simulation
MPF	Main Program File (NC parts program)
MPI	Multi-Point Interface
MSD	Main Spindle Drive
NC	Numerical Control
NCK	Numerical Control Kernel: Numerical kernel with block preparation, traversing range etc.
NCU	Numerical Control Unit
NURBS	Non-Uniform Rational B-Spline
O	Output -> UI
OEM	Original Equipment Manufacturer
OI	Operator Interface
OP	Operator Panel
OPI	Operator Panel Interface
PC	Personal Computer
PCMCIA	Personal Computer Memory Card International Association (interface convention)
PG	Programming device

© Siemens AG, 2002. All rights reserved

PLC	Programmable Logic Controller
PRT	PRogram Test
RAM	Random Access Memory (can be read and written)
RISC	Reduced Instruction Set Computer
ROV	Rapid OVerride
RPA	R Parameter Active: Memory area in the NCK for R parameter numbers
RTS	Request To Send: Control signal from serial data interfaces
SBL	Single BLock
SD	Setting Date
SEA	Setting Data Active: Memory area in the NCK for setting data
SKP	Skip Block
SM	Signal Module
SPF	SubProgram File
SSI	Serial Synchronous Interface
SW	Software
T	Tool
TC	Tool Compensation
TEA	Testing Data Active: refers to machine data
TO	Tool Offset
TOA	Tool Offset Active: Memory area for tool offsets

© Siemens AG, 2002. All rights reserved

TRANSMIT	Transform Milling into Turning: Coordinate conversion on turning machines for milling
UI	User Interface
WO	Work Offset -> Zero Offset
WOA	Work Offset Active -> ZOP
ZO	Zero Offset -> Work Offset
ZOA	Zero Offset Active: Memory area for zero (work) offsets -> WOA

© Siemens AG, 2002. All rights reserved

B References

General Documentation

/BU/ SINUMERIK 840D/840Di/810D/802S, C, D
Ordering Information
Catalog NC 60
Order No: E86060-K4460-A101-A9-7600

/IKPI/ **Catalog IK PI 2000**
Industrial Communication and Field Devices
Catalog ST 70
Order No. of bound edition: E86060-K6710-A101-A9-7600
Order No. of single-sheet edition: E86060-K6710-A100-A9-7600

/ST7/ **SIMATIC**
SIMATIC S7 Programmable Logic Controllers
Catalog ST 70
Order No.: E86060-K4670-A111-A3

/Z/ SINUMERIK, SIROTEC, SIMODRIVE
Accessories and Equipment for Special-Purpose Machines
Catalog NC Z
Order No.: E86060-K4490-A001-A8-7600

Electronic Documentation

/CD1/ The SINUMERIK system
DOC ON CD (11.02 Edition)
(with all SINUMERIK 840D/840Di/810D/802 and
SIMODRIVE publications)
Order No.: 6FC5 298-6CA00-0BG3

© Siemens AG, 2002. All rights reserved

User Documentation

/AUK/	SINUMERIK 840D/810D **AutoTurn Short Operating Guide** Order No.: 6FC5 298-4AA30-0BP3	(11.02 Edition)
/AUP/	SINUMERIK 840D/810D **AutoTurn Graphic Programming System** Operator's Guide Programming/Setup Order No.: 6FC5 298-4AA40-0BP3	(02.02 Edition)
/BA/	SINUMERIK 840D/810D **Operator's Guide MMC** Order No.: 6FC5 298-6AA00-0BP0	(10.00 Edition)
/BAD/	SINUMERIK 840D/840Di/810D **Operator's Guide HMI Advanced** Order No.: 6FC5 298-6AF00-0BP2	(11.02 Edition)
/BEM/	SINUMERIK 840D/810D **Operator's Guide HMI Embedded** Order No.: 6FC5 298-6AC00-0BP2	(11.02 Edition)
/BAH/	SINUMERIK 840D/840Di/810D **Operator's Guide HT 6 (new HPU)** Order No.: 6FC5 298-0AD60-0BP2	(06.02 Edition)
/BAK/	SINUMERIK 840D/840Di/810D **Short Operating Guide** Order No.: 6FC5 298-6AA10-0BP0	(02.01 Edition)
/BAM/	SINUMERIK 810D/840D **Operator's Guide ManualTurn** Order No.: 6FC5 298-6AD00-0BP1	(08.02 Edition)
/BAS/	SINUMERIK 840D/810D **Operator's Guide ShopMill** Order No.: 6FC5 298-6AD10-0BP1	(11.02 Edition)
/BAT/	SINUMERIK 840D/810D **Operator's Guide ShopTurn** Order No.: 6FC5 298-6AD50-0BP2	(03.03 Edition)
/BNM/	SINUMERIK 840D/840Di/810D **User's Guide Measuring Cycles** Order No.: 6FC5 298-6AA70-0BP2	(11.02 Edition)
/CAD/	SINUMERIK 840D/840Di/810D **Operator's Guide CAD Reader** Order No.: (included in the Online Help)	(03.02 Edition)

© Siemens AG, 2002. All rights reserved

/DA/	SINUMERIK 840D/840Di/810D **Diagnostics Guide** Order No.: 6FC5 298-6AA20-0BP3	(11.02 Edition)
/KAM/	SINUMERIK 840D/810D **Short Guide ManualTurn** Order No.: 6FC5 298-5AD40-0BP0	(04.01 Edition)
/KAS/	SINUMERIK 840D/810D **Short Guide ShopMill** Order No.: 6FC5 298-5AD30-0BP0	(04.01 Edition)
/KAT/	SINUMERIK 840D/810D **Short Guide ShopTurn** Order No.: 6FC5 298-6AF20-0BP0	(07.01 Edition)
/PG/	SINUMERIK 840D/840Di/810D **Programming Guide, Fundamentals** Order No.: 6FC5 298-6AB00-0BP2	(11.02 Edition)
/PGA/	SINUMERIK 840D/840Di/810D **Programming Guide, Advanced** Order No.: 6FC5 298-6AB10-0BP2	(11.02 Edition)
/PGK/	SINUMERIK 840D/840Di/810D **Short Guide, Programming** Order No.: 6FC5 298-6AB30-0BP1	(02.01 Edition)
/PGM/	SINUMERIK 840D/840Di/810D **Programming Guide ISO Milling** Order No.: 6FC5 298-6AC20-0BP2	(11.02 Edition)
/PGT/	SINUMERIK 840D/840Di/810D **Programming Guide ISO Turning** Order No.: 6FC5 298-6AC10-0BP2	(11.02 Edition)
/PGZ/	SINUMERIK 840D/840Di/810D **Programming Guide Cycles** Order No.: 6FC5 298-6AB40-0BP2	(11.02 Edition)
/PI/	**PCIN 4.4** Software for Data Transfer to/from MMC Module Order No.: 6FX2 060-4AA00-4XB0 (German, English, French) Order from: WK Fürth	
/SYI/	SINUMERIK 840Di **System Overview** Order No.: 6FC5 298-6AE40-0BP0	(02.01 Edition)

© Siemens AG, 2002. All rights reserved

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D
SIMODRIVE 611D
Lists (11.02 Edition)
Order No.: 6FC5 297-6AB70-0BP3

b) Hardware

/BH/ SINUMERIK 840D/840Di/810D
Operator Components Manual (HW) (11.02 Edition)
Order No.: 6FC5 297-6AA50-0BP2

/BHA/ SIMODRIVE **Sensor**
Absolute Position Sensor with Profibus DP
User's Guide (HW) (02.99 Edition)
Order No.: 6SN1 197-0AB10-0YP1

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE
EMC Installation Guide (06.99 Edition)
Planning Guide (HW)
Order No.: 6FC5 297-0AD30-0BP1

/GHA/ **ADI4 – Analog Drive Interface for 4 Axes** (09.02 Edition)
Manual
Order No.: 6FC5 297-0BA01-0BP0

/PHC/ SINUMERIK 810D
Configuring Manual (HW) (11.02 Edition)
Order No.: 6FC5 297-6AD10-0BP1

/PHD/ SINUMERIK 840D
Configuring Manual NCU 561.2-573.4 (HW) (10.02 Edition)
Order No.: 6FC5 297-6AC10-0BP2

/PMH/ SIMODRIVE **Sensor**
Hollow-Shaft Measuring System SIMAG H
Configuring/Installation Guide (HW) (07.02 Edition)
Order No.: 6SN1197-0AB30-0BP1

© Siemens AG, 2002. All rights reserved

c) Software

/FB1/ SINUMERIK 840D/840Di/810D/FM-NC
Description of Functions Basic Machine (Part 1) (11.02 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC20-0BP2

A2	Various Interface Signals
A3	Axis Monitoring, Protection Zones
B1	Continuous Path Mode, Exact Stop and Look Ahead
B2	Acceleration
D1	Diagnostic Tools
D2	Interactive Programming
F1	Travel to Fixed stop
G2	Velocities, Setpoint/Actual Value Systems, Closed-Loop Control
H2	Output of Auxiliary Functions to PLC
K1	Mode Group, Channels, Program Operation
K2	Axes, Coordinate Systems, Frames, Actual Value System for Workpiece, External Zero Offset
K4	Communication
N2	EMERGENCY STOP
P1	Transverse Axes
P3	Basic PLC Program
R1	Reference Point Approach
S1	Spindles
V1	Feeds
W1	Tool Compensation

/FB2/ SINUMERIK 840D/840Di/810D
Description of Functions, Extended Functions (Part 2) (11.02 Edition)
including FM-NC: Turning, Stepping Motor
(the various sections are listed below)
Order No.: 6FC5 297-6AC30-0BP2

A4	Digital and Analog NCK I/Os
B3	Several Operator Panels and NCUs
B4	Operation via PC/PG
F3	Remote Diagnostics
H1	Jog with/without Handwheel
K3	Compensations
K5	Mode Groups, Channels, Axis Exchange
L1	FM-NC Local Bus
M1	Kinematic Transformation
M5	Measurements
N3	Software Cams, Position Switching Signals
N4	Punching and Nibbling
P2	Positioning Axes
P5	Oscillation
R2	Rotary Axes
S3	Synchronous Spindles
S5	Synchronized Actions (up to and including SW 3/subsequently /FBSY/)
S6	Stepper Motor Control
S7	Memory Configuration
T1	Indexing Axes
W3	Tool Change
W4	Grinding

© Siemens AG, 2002. All rights reserved

/FB3/ SINUMERIK 840D/840Di/810D
Description of Functions, Special Functions (Part 3) (11.02 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC80-0BP2

F2 3 to 5 Axis Transformation
G1 Gantry Axes
G3 Cycle Times
K6 Contour Tunnel Monitoring
M3 Coupled Axes and ESR
S8 Constant Workpiece Speed for Centerless Grinding
T3 Tangential Control
TE1 Clearance Control
TE2 Analog Axis
TE3 Master/Slave for Drives
TE4 Transformation Package Handling
TE1 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Clock-Independent Path-Synchronous Switching Signal Output
V2 Preprocessing
W5 3D Tool Radius Compensation

/FBA/ SIMODRIVE 611D/SINUMERIK 840D/810D
Description of Functions, Drive Functions (11.02 Edition)
(the various sections are listed below)
Order No.: 6SN1 197-0AA80-1BP0

DB1 Operational Messages/Alarm Reactions
DD1 Diagnostic Functions
DD1 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
DL1 Linear Motor MD
DM1 Calculation of Motor/Power Section Parameters and Controller Data
DS1 Current Control Loop
DÜ1 Monitors/Limitations

/FBAN/ SINUMERIK 840D/SIMODRIVE 611 DIGITAL
Description of Functions **ANA MODULE** (02.00 Edition)
Order No.: 6SN1 197-0AB80-0BP0

/FBD/ SINUMERIK 840D
Description of Functions **Digitizing** (07.99 Edition)
Order No.: 6FC5 297-4AC50-0BP0

DI1 Start-up
DI2 Scanning with Tactile Sensors (scancad scan)
DI2 Scanning with Lasers (scancad laser)
DI4 Milling Program Generation (scancad mill)

© Siemens AG, 2002. All rights reserved

/FBDN/ SINUMERIK 840D/810D
IT Solutions
NC Data Management Server (DNC NT-2000) (01.02 Edition)
Description of Functions
Order No.: 6FC5 297-5AE50-0BP2

/FBDT/ SINUMERIK 840D/840Di/810D
IT Solutions
Data Transfer via Network (09.01 Edition)
SinDNC-FS, SinDNC-DB
Version 5.1
Description of Functions
Order No.: 6FC5 297-5AE70-0BP0

/FBFA/ SINUMERIK 840D/840Di/810D
ISO Dialects for SINUMERIK (11.02 Edition)
Description of Functions
Order No.: 6FC5 297-6AE10-0BP3

/FBFE/ SINUMERIK 840D/810D
Description of Functions **Remote Diagnosis** (11.02 Edition)
Order No.: 6FC5 297-0AF00-0BP2

/FBH/ SINUMERIK 840D/840Di/810D
HMI Configuration Package (11.02 Edition)
Order No.: (is supplied with the software)

Part 1 User's Guide
Part 2 Description of Functions

/FBHLA/ SINUMERIK 840D/SIMODRIVE 611 digital
Description of Functions **HLA MODULE** (11.02 Edition)
Order No.: 6SN1 197-0AB60-0BP3

/FBMA/ SINUMERIK 840D/810D
Description of Functions **ManualTurn** (08.02 Edition)
Order No.: 6FC5 297-5AD50-0BP2

/FBO/ SINUMERIK 840D/810D
Configuring OP 030 Operator Interface (09.01 Edition)
Description of Functions
(the various sections are listed below)
Order No.: 6FC5 297-6AC40-0BP0

BA Operator's Guide
EU Development Environment (Configuring Package)
PSE Introduction to Configuring of Operator Interface
IK Screen Kit: Software Update And Configuration

/FBP/ SINUMERIK 840D
Description of Functions **C-PLC Programming** (03.96 Edition)
Order No.: 6FC5 297-3AB60-0BP0

© Siemens AG, 2002. All rights reserved

/FBR/	SINUMERIK 840D/810D IT Solutions **Computer Link** (SinCOM) Description of Functions Order No.: 6FC5 297-6AD60-0BP0 NFL Host Computer Interface NPL PLC/NCK Interface	(09.01 Edition)
/FBSI/	SINUMERIK 840D/SIMODRIVE Description of Functions **SINUMERIK Safety Integrated** Order No.: 6FC5 297-6AB80-0BP1	(07.02 Edition)
/FBSP/	SINUMERIK 840D/810D Description of Functions **ShopMill** Order No.: 6FC5 297-6AD80-0BP1	(11.02 Edition)
/FBST/	**SIMATIC** Description of Functions **FM STEPDRIVE/SIMOSTEP** Order No.: 6SN1 197-0AA70-0YP4	(01.01 Edition)
/FBSY/	SINUMERIK 840D/810D Description of Functions **Synchronized Actions** Order No.: 6FC5 297-6AD40-0BP2	(10.02 Edition)
/FBT/	SINUMERIK 840D/810D Description of Functions **ShopTurn** Order No.: 6FC5 297-6AD70-0BP2	(03.03 Edition)
/FBTC/	SINUMERIK 840D/810D IT Solutions **SINUMERIK Tool Data Communication SinTDC** Description of Functions Order No.: 6FC5 297-5AF30-0BP0	(01.02 Edition)
/FBTD/	SINUMERIK 840D/810D IT Solutions **Tool Data Information System** (SinTDI) with Online Help Description of Functions Order No.: 6FC5 297-6AE00-0BP0	(03.01 Edition)
/FBU/	**SIMODRIVE 611 universal/universal E** Closed-Loop Control Component for Speed Control and Positioning Description of Functions Order No.: 6SN1 197-0AB20-0BP6	(08.02 Edition)
/FBW/	SINUMERIK 840D/810D Description of Functions **Tool Management** Order No.: 6FC5 297-6AC60-0BP1	(10.02 Edition)
/FBWI/	SINUMERIK 840D/840Di/810D Description of Functions **WinTPM** Order No.: The document is an integral part of the software	(02.02 Edition)

© Siemens AG, 2002. All rights reserved

/HBA/ SINUMERIK 840D/840Di/810D
Manual @Event (03.02 Edition)
Order No.: 6AU1900-0CL20-0BA0

/HBI/ SINUMERIK 840Di
Manual (09.02 Edition)
Order No.: 6FC5 297-6AE60-0BP1

/INC/ SINUMERIK 840D/840Di/810D
Commissioning Tool **SINUMERIK SinuCOM NC** (09.02 Edition)
System Description
Order No.: (an integral part of the online help for the start-up tool)

/PAP/ SIMODRIVE **Sensor**
Absolute value encoder with Profibus-DP (02.99 Edition)
User's Guide
Order No.: 6SN1197-0AB10-0YP1

/PFK/ **SIMODRIVE**
Planning Guide
1FT5/1FT6/1FK6 Motors (12.02 Edition)
AC Servo Motors for Feed and Main Spindle Drives
Order No.: 6SN1 197-0AC20-0BP0

/PJE/ SINUMERIK 840D/810D
Configuration Package HMI Embedded (08.01 Edition)
Description of Functions: Software Update, Configuration, Installation
Order No.: 6FC5 297-6EA10-0BP0
(the PS Configuring Syntax document is supplied with the software and is also available as a pdf file)

/PJFE/ **SIMODRIVE**
Planning Guide (11.02 Edition)
Synchronous Integrated Motors 1FE1
AC Motors for Main Spindle Drives
Order No.: 6SN1 197-0AC00-0BP3

/PJLM/ SIMODRIVE
Planning Guide **Linear Motors 1FN1, 1FN3** (06.02 Edition)

ALL General Information about Linear Motors
1FN1 Three-Phase AC Linear Motor 1FN1
1FN3 Three-Phase AC Linear Motor 1FN3
CON Connections
Order No.: 6SN1 197-0AB70-0BP4

/PJM/ **SIMODRIVE**
Planning Guide **Motors**
Three-Phase AC Motors for Feed and Main Spindle Drives (11.00 Edition)
Order No.: 6SN1 197-0AA20-0BP5

© Siemens AG, 2002. All rights reserved

/PJTM/	**SIMODRIVE** Planning Guide **Integrated Torque Motors 1FW6** Order No.: 6SN1 197-0AD00-0BP0	(08.02 Edition)
/PJU/	**SIMODRIVE 611** Planning Guide **Inverters** Order No.: 6SN1 197-0AA00-0BP6	(08.02 Edition)
/PMS/	**SIMODRIVE** Planning Guide **ECO Motor Spindle** for Main Spindle Drives Order No.: 6SN1 197-0AD04-0BP0	(04.02 Edition)
/POS1/	**SIMODRIVE POSMO A** User's Guide Distributed Positioning Motor on PROFIBUS DP Order No.: 6SN2 197-0AA00-0BP3	(08.02 Edition)
/POS2/	**SIMODRIVE POSMO A** Installation Guide (included in every POSMO A)	
/POS3/	**SIMODRIVE POSMO SI/CD/CA** Distributed Servo Drive Systems, User's Guide Order No.: 6SN2 197-0AA20-0BP3	(08.02 Edition)
/PPH/	**SIMODRIVE** Planning Guide **1PH2/1PH4/1PH7 Motors** AC Induction Motors for Main Spindle Drives Order No.: 6SN1 197-0AC60-0BP0	(12.01 Edition)
/PPM/	**SIMODRIVE** Planning Guide **Hollow-Shaft Motors** Hollow-Shaft Motors for Main Spindle Drives 1PM4 and 1PM6 Order No.: 6SN1 197-0AD03-0BP0	(10.01 Edition)
/S7H/	**SIMATIC S7-300** - Manual: CPU Data (Hardware) - Reference Manual: Module Data - Manual: Technological Functions - Installation Manual Order No.: 6ES7 398-8AA03-8AA0	(2002 Edition)
/S7HT/	**SIMATIC S7-300** Manual: STEP 7, Fundamentals, V. 3.1 Order No.: 6ES7 810-4CA02-8AA0	(03.97 Edition)
/S7HR/	**SIMATIC S7-300** Manual: STEP 7, Reference Manuals, V. 3.1 Order No.: 6ES7 810-4CA02-8AR0	(03.97 Edition)

© Siemens AG, 2002. All rights reserved

/S7S/	**SIMATIC S7-300** **FM 353** Positioning Module for Stepper Drives Order in conjunction with Configuring Package	(04.97 Edition)
/S7L/	**SIMATIC S7-300** **FM 354** Positioning Module for Servo Drives Order in conjunction with Configuring Package	(04.97 Edition)
/S7M/	**SIMATIC S7-300** **FM 357.2** Multimodule for Servo or Stepper Drives Order in conjunction with Configuring Package	(01.01 Edition)
/SP/	**SIMODRIVE 611-A/611-D** **SimoPro 3.1** Program for Configuring Machine Tool Drives Order No.: 6SC6 111-6PC00-0AA□ Order from: WK Fürth	

d) Installation and Start-Up

/IAA/	**SIMODRIVE 611A** **Installation and Start-Up Guide** Order No.: 6SN 1197-0AA60-0BP6	(10.00 Edition)
/IAC/	SINUMERIK 810D **Installation and Start-Up Guide** (including description of SIMODRIVE 611D start-up software) Order No.: 6FC5 297-6AD20-0BP0	(03.02 Edition)
/IAD/	SINUMERIK 840D/SIMODRIVE 611D **Installation and Start-Up Guide** (including description of SIMODRIVE 611D start-up software) Order No.: 6FC5 297-6AB10-0BP2	(11.02 Edition)
/IAM/	SINUMERIK 840D/840Di/810D **HMI/MMC Installation and Start-Up Guide** Order No.: 6FC5 297-6AE20-0BP2	(11.02 Edition)

AE1	Updates/Supplements
BE1	Expanding the Operator Interface
HE1	Online Help
IM2	Starting-Up HMI Embedded
IM4	Starting-Up HMI Advanced
TX1	Creating Foreign Language Texts

© Siemens AG, 2002. All rights reserved

© Siemens AG, 2002. All rights reserved

To

SIEMENS AG

A&D MC BMS

P.O. Box 3180

D-91050 Erlangen, Germany

Tel. ++49-(0)180-5050-222 (Hotline)
Fax ++49-(0)9131-98-2176 (Documentation)
email: motioncontrol.docu@erlf.siemens.de

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 840D/840Di/810D

User Documentation

From

Name

Company/Dept.

Address

Tel.: /

Fax: /

Diagnostics Guide

Order No.: 6FC5 298-6AA20-0BP3
Edition: 11.02

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections

Overview of SINUMERIK 840D/840Di/810D Documentation (11.2002)

General Documentation

- *SINUMERIK* 840D/810D — Brochure
- *SINUMERIK* 840D/840Di/ 810D/ — Catalog Ordering Info. NC 60 ***)**
- *SINUMERIK SIROTEC SIMODRIVE* Accessories — Catalog Accessories NC-Z

User Documentation

- *SINUMERIK* 840D/810D/ FM-NC — **AutoTurn**
  – Short Guide
  – Programming/ Setup
- *SINUMERIK* 840D/840Di/ 810D — Operator's Guide **– HT 6**
- *SINUMERIK* 840D/840Di/ 810D — **Diagnostics Guide *)**
- *SINUMERIK* 840D/840Di/ 810D — **Operator's Guide *)**
  – Short Guide
  – HMI Embedded
  – HMI Advanced

User Documentation

- *SINUMERIK* 840D/840Di/ 810D — **Program. Guide**
  – Short Guide
  – Fundamentals ***)**
  – Advanced ***)**
  – Cycles
  – Measuring Cycles
  – ISO Turning/Milling
- *SINUMERIK* 840D/810D — Operator's Guide
  – **ManualTurn**
  – Short Guide ManualTurn
  – **ShopMill**
  – Short Guide ShopMill
  – **ShopTurn**
  – Short Guide ShopTurn
- *SINUMERIK* 840Di — **System Overview**

Manufacturer/Service Documentation

- *SINUMERIK* — **Configuring** (HW) ***)**
  – 810D
  – 840D
- *SINUMERIK* 840D/840Di/ 810D — **Operator Components** (HW) ***)**
- *SINUMERIK* 840D/810D — Description of Functions **– ManualTurn** **– ShopMill** **– ShopTurn**
- *SINUMERIK* 840D/840Di/ 810D — Description of Functions **Synchronized Actions**

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* 611D 840D/810D — Description of Functions **Drive Functions *)**
- *SINUMERIK* 840D/840Di/ 810D — **Description of Functions**
  – Basic Machine ***)**
  – Extended Functions
  – Special Functions
- *SINUMERIK* 840D/840Di/ 810D — Description of Functions **Tool Management**
- *SINUMERIK* 840D/810D — **Configuring Kit** HMI Embedded
- *SINUMERIK* 840D/810D — Description of Functions Operator Interface **OP 030**
- *SINUMERIK* 840D/810D — **IT Solutions**
  – Computer Link
  – Tool Data Information System
  – NC Data Management
  – NC Data Transfer
  – Tool Data Communication

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* — Description of Functions SINUMERIK **Safety Integrated**
- *SINUMERIK* 840D — Description of Functions **Digitizing**
- *SINUMERIK SIMODRIVE* — **Installation & Start-Up Guide *)**
  – 810D
  – 840D/611D
  – HMI
- *SINUMERIK SIMODRIVE* 840D/840Di 810D 611D — **Lists *)**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **Linear Motor**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **– Hydraulics Module** **– Analog Module**
- *SINUMERIK SIMODRIVE SIROTEC* — **EMC** Guidelines

Electronic Documentation

- *SINUMERIK SIMODRIVE* 840D/840Di/ 810D 611, Motors — **DOC ON CD *)** The SINUMERIK System

Manufacturer/Service Documentation

- *SINUMERIK* 840D/840Di/ 810D — Description of Functions **ISO Dialects** for SINUMERIK
- *SINUMERIK* 840Di — **Manual** (HW + Installation and Start-Up)
- *SINUMERIK* 840D/810D — Description of Functions **Remote Diagnosis**
- *SINUMERIK* 840D/840Di/ 810D — **Manual @ Event**

***)** These documents are a minimum requirement

Siemens AG
Automation & Drives
Motion Control Systems
P.O. Box 3180, D-91050 Erlangen
Germany

www.ad.siemens.de

© Siemens AG, 2002
Subject to change without prior notice
Order No: 6FC5 298-6AA20-0BP3

Printed in Germany