SIEMENS

SINUMERIK 802D sl

Diagnostics Manual

Valid for

Control system	*Software version*
SINUMERIK 802D sl	1.4

06/2007 Edition

SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

A .... New documentation.

B .... Unrevised reprint with new order number

C .... Revised edition with new status

Edition	Order No.	Remark
12/2004	6FC5398-2CP10-0BA0	**O**
06/2007	6FC5398-2CP10-1BA0	**C**

Registered trademarks
All designations with the trademark symbol ® are registered trademarks of Siemens AG. Other designations in this documentation may be trademarks whose use by third parties for their own purposes may infringe the rights of the owner.

Disclaimer
We have checked the contents of this document for agreement with the hardware and software described. Nevertheless, as deviations cannot be precluded entirely, we cannot guarantee complete accuracy of the information contained herein. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition.

Copyright © Siemens AG 1995 - 2007
Order No.6FC5398-2CP10-1BA0
Siemens-Aktiengesellschaft
Subject to change without prior notice

Preface

Structure of the documentation

The SINUMERIK documentation is organized in 3 parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

An overview of publications, which is updated monthly and also provides information about the language versions available, can be found on the Internet at:

http://www.siemens.com/motioncontrol
Select "Support" -> "Technical Documentation" ->"Overview of Publications"

The Internet version of the DOConCD (DOConWEB) is available at:
http://www.automation.siemens.com/doconweb

Information about training courses and FAQs (Frequently Asked Questions) can be found in internet under:

http://www.siemens.com/motioncontrol under menu option "Support".

Target group

This publication is intended for project engineers, commissioners, machine operators and service and maintenance personnel.

Benefits

The Diagnostics Manual enables the intended target group to evaluate error and fault indications and to respond accordingly.

It allows the operator at the machine tools:

- 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.

" Siemens AG 2007 All Rights Reserved

Standard scope

This Diagnostics Manual describes only the functionality of the standard version. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

Other functions not described in this documentation might be executable in the control. However, no claim can be made regarding the availability of these functions when the equipment is first supplied or in the event of servicing.

Further, for the sake of simplicity, this documentation does not contain all detailed information about all types of the product and cannot cover every conceivable case of installation, operation or maintenance.

Technical Support

If you have any questions, please contact the following hotline:

	Europa / Afrika
Phone	+49 180 5050 - 222
Fax	+49 180 5050 - 223
Internet	http://www.siemens.de/automation/support-request

	Amerika
Phone	+1 423 262 2522
Fax	+1 423 262 2200
E-Mail	mailto:techsupport.sea@siemens.com

	Asien / Pazifik
Phone	+86 1064 719 990
Fax	+86 1064 747 474
E-Mail	mailto:adsupport.asia@siemens.com

Note

Country-specific telephone numbers for technical support are provided under the following Internet address:

http://www.siemens.com/automation/service&support.

" Siemens AG 2007 All Rights Reserved

Questions regarding documentation

If you have any queries (suggestions, corrections) in relation to this documentation, please fax or email us:

Fax	+49 9131 98 - 63315
E-mail	mailto:docu.motioncontrol@siemens.com

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

SINUMERIK Internet address

http://www.siemens.com/sinumerik

EC Declaration of Conformity

The EC Declaration of Conformity for the EMC Directive can be found/obtained

on the Internet:

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

under the Product/Order No. 15257461

at the relevant branch office of the A&D MC group of Siemens AG

Subject matter of this manual

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, PROFIBUS, the cycles and the PLC.

Additional alarms from the HMI (Human Machine Interface) area may occur as well. These alarms are displayed on the operator panel in the form of self-explanatory text. They are not subject of this Diagnostics Guide.

For special situations in conjunction with the integrated PLC, please refer to the SIMATIC S7-200 documentation.

" Siemens AG 2007 All Rights Reserved

Sorting

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

Structure of alarm description

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

- Explanation
- Response
- Remedy
- Program continuation

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

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

Safety

Danger

Please check the situation in the plant on the basis of the description of the active alarm(s). Eliminate the causes for the occurrence of the alarms and acknowledge in the manner indicated. Failure to observe this warning will place your machine, workpiece, stored settings and possibly even your own safety at risk.

NCK alarms

Table -1 Number ranges of the alarm numbers

002 000 - 009 999	General alarms
010 000 - 019 999	ISO 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

" Siemens AG 2007 All Rights Reserved

PROFIBUS alarms

Table -2 Number ranges of the message numbers, continued

380000 - 380502	PROFIBUS alarms

PLC alarms/messages

Table -3 Number ranges of the alarm numbers, continued

400000 - 400015	General alarms

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

Safety information

This manual contains information which you should observe to ensure your own personal safety as well as to protect the product and connected equipment. Notices referring to your personal safety are highlighted in the manual by a safety alert symbol; notices referring to property damage only have no safety alert symbol. These notices shown below are graded according to the degree of danger.

Danger

Indicates that death or severe personal injury will result if proper precautions are not taken.

t

Warning

means that there can be severe physical injury or even death if the corresponding safety measures are not followed.

Caution

means that there can be slight physical injury if the corresponding safety measures are not followed.

Caution

means that there can be damage to property if the corresponding safety measures are not followed.

" Siemens AG 2007 All Rights Reserved

NOTICE

indicates that an undesirable result or state may occur if the corresponding instruction is not followed.

In the event of a number of levels of danger prevailing simultaneously, the warning corresponding to the highest level of danger is always used. A warning notice accompanied by a safety alert symbol indicating a risk of bodily injury can also indicate a risk of property damage.

Qualified persons

The associated device/system must only be set up and operated using this documentation. The device/system must be commissioned and operated by qualified personnel only. Qualified personnel as defined under the safety guidelines in this documentation are those who are authorized to start up, earth and label units, systems and circuits in accordance with the relevant safety standards.

" Siemens AG 2007 All Rights Reserved

Table of Contents

1

" Siemens AG 2007 All Rights Reserved

" Siemens AG 2007 All Rights Reserved

Overview of system error alarms

1

System fault

Alarms with alarm number 1xxx are system errors that indicate internal error states. The internal error number transmitted provides the developer important information with regard to the error cause and the error location.

These system error alarms are not described in detail! If they occur with the supplied control systems at all, please contact the following hotline specifying

- the alarm number
- the alarm text and
- the internal system error number contained therein.

The indications regarding your country-specific hotline are provided in the foreword under Technical Support.

" Siemens AG 2007 All Rights Reserved

" Siemens AG 2007 All Rights Reserved

Overview of alarms 2

2.1 NCK alarms/ISO alarms

2000 PLC sign-of-life monitoring

Definitions: The PLC must give a sign of life within the specified period. An alarm is issued if this does not happen.

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

Remedy: This alarm also occurs after a PLC Stop.
(PLC Stop with programming tool,
PLC Stop from commissioning switch,
PLC Stop from alarm)
If none of these cases apply, contact the hotline stated at the beginning of the document and state the number of the operating system error.

Program Continuation: Switch control OFF - ON.

2001 PLC has not started up

Definitions: The PLC must give at least one sign of life within the specified period after Power On.

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

Remedy: Contact the hotline stated at the beginning of the document.

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 2.7-2.9 V (nominal voltage of the battery is 3.0-3.1 V at 950 mAh).

Reaction: 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 2.4-2.6 V if the RAMs to be buffered take up a lot of current.

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

" Siemens AG 2007 All Rights Reserved

2101 NCK battery alarm

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

Reaction: 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.

2102 NCK battery alarm

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

Reaction: NC not ready.
The NC switches to follow-up mode.
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 1/2 AA, 850 mAh, min. 3.2 V). 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 degrees C +/- 2.5 degrees C.

Reaction: Alarm display.

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

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

2120 NCK fan alarm

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

Reaction: 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 2007 All Rights Reserved

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

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

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

Remedy: Please inform the authorized personnel/service department. Check the encoder and cable for short-circuits (the fault must not recur when you remove the cable). Check the voltage supply.

Program Continuation: Switch control OFF - ON.

2140 The actual service switch position forces the 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.

Reaction: NC not ready.
Interface signals are set.
Alarm display.

Remedy: Reset initialization switch to zero.

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

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 (RS422 board plugged into the NCU) for communication with the digitizing unit. This module was not found when booting.

Reaction: Interface signals are set.
Alarm display.

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

Program Continuation: Switch control OFF - ON.

2194 Link axis active and $MN_MM_SERVO_FIFO_SIZE != 3

Definitions: At least one axis is to be distributed via NCU link, then the machine data $MN_MM_SERVO_FIFO_SIZE must be 3. Occurs only with an NCU link system.

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

Remedy: Set $MN_SERVO_FIFO_SIZE to 3.

Program Continuation: Switch control OFF - ON.

2200 Channel %1 fast punching/nibbling not possible in several channels

Parameters: %1 = Channel number

Definitions: An attempt was made to activate fast nibbling or punching in a channel while it has already been active in another channel. Fast punching and nibbling is only possible simultaneously in the same channel.

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

Remedy: Fast nibbling and punching simultaneously in 1 channel only.

" Siemens AG 2007 All Rights Reserved

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

2900 Reboot is delayed

Definitions: This alarm indicates a delayed reboot.
The alarm occurs only when the reboot is carried out by the HMI and the MD 10088 REBOOT_DELAY_TIME is set higher than zero.
The alarm can be suppressed with SUPPRESS_ALARM_MASK BIT 20.

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

Remedy: See MD 10088 REBOOT_DELAY_TIME and
MD 11410 SUPPRESS_ALARM_MASK

Program Continuation: Switch control OFF - ON.

3000 Emergency stop

Definitions: The EMERGENCY STOP request is present at the NC/PLC interface (V 26000000.1).

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

Remedy: Check that an EMERGENCY STOP cam was not approached and that no EMERGENCY STOP button was pressed. Check PLC user program.
Corrected the cause of EMERGENCY STOP and acknowledge it via the NC/PLC interface (V 26000000.2).

Program Continuation: Teileprogramm neu starten.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.

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

Remedy: No remedy required

Program Continuation: Teileprogramm neu starten.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.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Please inform the authorized personnel/service department.
Configure MD 20070 AXCONF_MACHAX_USED for the axis assignment in the channel without gaps; i.e. with increasing channel indices, a machine axis must be assigned until a zero is entered for the first time (no machine axis).
All MD with higher indices must then contain a zero (0).
The order of the machine axis numbers is not relevant.

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 via MD 20070 AXCONF_MACHAX_USED [kx]=m may be declared by means of MD 20050 AXCONF_GEOAX_ASSIGN_TAB [gx]=k.
gx ...geometry axis index:k ...channel axis number
kx ...channel axis indexm ...machine axis No.

Reaction: NC not ready.
Mode group not ready, also effective for single axes
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 and correct MD 20050 AXCONF_GEOAX_ASSIGN TAB and
MD 20070 AXCONF_MACHAX_USED.

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.

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

Remedy: Correct MD 20050 AXCONF_GEOAX_ASSIGN_TAB

Program Continuation: Switch control OFF - ON.

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 not 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

" Siemens AG 2007 All Rights Reserved

- $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.

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

Remedy: Modify $MC_AXCONF_MACHAX_USED.

Program Continuation: Switch control OFF - ON.

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

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

Definitions: When defining names for machine axes, one of the following syntax rules for the identifier to be entered has been violated:
1.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.
2.The identifier must begin with 2 arbitrary capital letters but not with $ (reserved).
3.The identifier must not be a vocabulary word of the NC language (e.g. SPOS).

Reaction: NC not ready.
Mode group not ready, also effective for single axes
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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:MD 10000 AXCONF_MACHAX_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.
The identifier must begin with 2 arbitrary capital letters but not with $ (reserved for system variable).
The identifier must not be a vocabulary word of the NC language (e.g. SPOS).

Reaction: NC not ready.
Mode group not ready, also effective for single axes
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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:MD 20060 AXCONF_GEOAX_ASSIGN_TAB
Channel axes:MD 10000 AXCONF_MACHAX_NAME_TAB

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

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

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

Definitions: The selected identifier is invalid. The following are valid identifiers:
• AX1 - AXn: Machine axis identifiers

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

Remedy: Use the correct identifier.

Program Continuation: Switch control OFF - ON.

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

Parameters: %1 = string: Identifier
%2 = string: MD identifier

Definitions: To determine the name in the channel-specific tables (arrays) for machine axes an identifier already existing in the control has been used.

Reaction: NC not ready.
Mode group not ready, also effective for single axes
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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: Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group.
Restart part program.

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

Parameters: %1 = channel number
%2 = string: 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.

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

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

Parameters: %1 = channel number
%2 = string: MD identifier
%3 = index: MD array index

" Siemens AG 2007 All Rights Reserved

Definitions: An axis identifier is expected for the displayed MD in accordance with the axis configuration in the AXCONF_MACHAX_USED and MD 20050 AXCONF_GEOAX_ASSIGN_TAB.

Reaction: NC not ready.
Mode group not ready, also effective for single axes
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 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 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 MD 20050 AXCONF_GEOAX_ASSIGN_TAB for the relevant geometry axis.

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 MD 20150 GCODE_RESET_VALUES or MD 20100 DIAMETER_AX_DEF, a facing axis identifier is expected at the specified location.

Reaction: 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.
Add the correct identifier.

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

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.

Reaction: NC not ready.
Mode group not ready, also effective for single axes
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 and correct the identifier used in the MD 10000 AXCONF_ MACHAX_NAME_TAB, MD 20080 AXCONF_CHANAX_NAME_TAB and/or MD 20050 AXCONF_GEOAX_NAME_TAB.

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

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

Parameters:

Definitions: Using the specified machine data %1 leads to a conflict with machine data %2.

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

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/vocabulary words can be reconfigured per machine data as long as you stay within the type range.
Type 1: "true" G codes:G02, G17, G33, G64, ...
Type 2: named G codes:CIP, TRANS, ...
Type 3: addresses which can be set:X, Y, I, J, K, MEAS

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

Remedy: Please inform the authorized personnel/service department.
Correct MD 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] = " "
deletes the function ROT from the control

Program Continuation: Switch control OFF - ON.

4060 Standard machine data loaded (%1, %2)

Parameters:

Definitions: Startup with defaults values by means of:
Operation (e.g. commissioning switch)
MD 11200 INIT_MD
Loss of remanent data
Startup operation with backup data, without saving current data

Reaction: Alarm display.

Remedy: After automatically loading the standard MDs, the individual MDs must be entered/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.

Reaction: Alarm display.

Remedy: Load specific machine data again.

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

" Siemens AG 2007 All Rights Reserved

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

Definitions: A possible inconsistency 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. 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 carried out via the "Save data" softkey on the HMI.

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

Remedy: Restart the control.

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.

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

Remedy: Start the control again.

Program Continuation: Switch control OFF - ON.

4070 Normalizing machine data has been changed

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 using scaling factors which can be entered (system-specific MD array 10230 SCALING_FACTORS USER_DEF[n] (n ... Index number 0 - 10) when the relevant 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:
MD 10220 SCALING_USER_DEF_MASK
MD 10230 SCALING_FACTORS_USER_DEF
MD 10240 SCALING_SYSTEM_IS_METRIC
MD 10250 SCALING_VALUE_INCH
MD 30300 IS_ROT_AX
After these data are changed, the NC must be restarted. Only then will the input of dependent data be performed correctly.

Reaction: 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 NC 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.

4071 Check the position of the encoder

Definitions: A machine data that affects the position of an absolute encoder has been changed. Please check the position values.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

" Siemens AG 2007 All Rights Reserved

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

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

Parameters: %1 = string: MD identifier
%2 = write-protection level of MD

Definitions: On executing a TOA file, 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 affected was not written.
This alarm is set only when access violation is detected for the first time.

Reaction: Alarm display.

Remedy: Set the required access level per password entry or delete the machine data concerned from the MD file.

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

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

Parameters: %1 = number of MD
%2 = access authorization set

Definitions: On executing a TOA file, an attempt has been made to write data with a higher protection level than the access authorization currently set in the control.
The data affected was not written.
This alarm is issued when acknowledging Alarm 4075. It can be cleared only with Power On.

Reaction: NC Start disable in this channel.
Alarm display.

Remedy: Set the required access level by means of keyswitch or password entry or delete the machine data concerned from the MD file 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 to many requested
%4 = Type of memory

Definitions: An attempt was made to enter a new value in the specified memory configuration machine data.
The change is not made, since it would result in the user memory being deleted. 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 MD18210 MM_USER_MEM_DYNAMIC.
"S" stands for static or buffered user memory (typically the part programs but also compensation data, R parameters and tool data). The capacity of this memory type is defined by the current memory expansion and the value in MD 18230 MM_USER_MEM_BUFFERED.

Reaction: Alarm display.

Remedy: If the modification was unintentional, simply continue.
The alarm has no negative effects.
The remedy depends on the access authorization and the current memory expansion of the NC.
The intended change is mot possible -> try again with a lower value. Observe how the value of the byte number changes.
Buy more memory? This option depends on the particular model used.
The NC user memory setting may be smaller than possible. The MDs (see above) can be altered using the relevant access authorization.

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

" Siemens AG 2007 All Rights Reserved

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.

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

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.

Reaction: NC Start disable in this channel.
Alarm display.

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) and/or MD 10080 SYSCLOCK_SAMPL_TIME_RATIO (dividing factor of the position control cycle for actual value acquisition) have been corrected. The sampling cycle to which the digital drive is synchronized (drive clock cycle) must be a multiple of 4, 8, 16 or 32 of 31.25µs.
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 31.25µs grid. If this requirement is unfeasible (e.g. because the system clock cycle is not a multiple of 31.25µs), the system clock cycle is automatically increased until the basic drive cycle is in a 31.25µs 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 4ms: 125µs step
- up to 8ms: 250µs step
- up to 16ms: 0.5ms step
- up to 32ms: 1ms step

Reaction: 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.

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.

Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

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.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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 cycle factor changed to %1

Parameters: %1 = String (new IPO cycle)
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 the SDB (MD 10050 SYSCLOCK_CYCLE_TIME).
Reaction: 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:
Definitions: The PLC cycle divisor was set to a value which was not an integral multiple of the IPO cycle divisor.
The divisor (MD 10 074 PLC_IPO_TIME_RATIO) has been increased.
Reaction: Alarm display.
Remedy: Adapt machine data.
Program Continuation: Clear alarm with the RESET key. Restart part program

4112 Servo cycle changed to %1 ms

Parameters: %1 = String (new servo cycle)
Definitions: MD 10060 POSCTRL_SYSCLOCK_TIME_RATIO has been modified because of the modified DP cycle in SDB1000 (10050 SYSCLOCK_CYCLE_TIME).
Reaction: Alarm display.
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)
Definitions: MD 10050 SYSCLOCK_CYCLE_TIME has been modified because of the modified DP cycle in the SDB.
Reaction: Alarm display.
Remedy: Machine data 10050 SYSCLOCK_CYCLE_TIME has been modified.
Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

4114 Error in DP cycle of SDB-Type-2000

Parameters: %1 = String (new PLC cycle)

Definitions: The DP cycle in the SDB contains an error and cannot be set.
The default value of MD 10050 SYSCLOCK_CYCLE_TIME is set.

Reaction: Alarm display.

Remedy: Correct the SDB.

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.

Reaction: 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:

Definitions: In MD MD 10715 M_NO_FCT_CYCLE, an M function that is occupied by the system has been specified for the configuration of the subprogram call and cannot be replaced by a subprogram call (M0 to M5, M17, M19, M30, M40 to M45, M70). When external language is active, M96 to M99 are also blocked

Reaction: 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 an M function which is not used by the system (M0 to M5, M17, M19, M30, M40 to M45 and M70) in MD 10715 M_NO_FCT_CYCLE.

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 MD 28400 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 MD 28402 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] <= (MD 28060 MM_IPO_BUFFER_SIZE + MD 28070 MM_NUM_BLOCKS_IN_PREP).
Alarm 4152 is issued if the limits are violated.

Reaction: 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 block length/display range within the permissible limits.

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

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

Parameters:

Definitions: %1=Channel number
An M function was specified in MD 20094 SPIND_RIGID_TAPPING_M_NR to configure the M function number for spindle switchover to axis mode. The M function number is assigned by the system and cannot be used for the switchover (M1 to M5,M17, M30, M40 to M45).

Reaction: 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 an M function which is not used by the system (M1 to M5, M17, M30, M40 to M45) in MD 20094 SPIND_RIGID_TAPPING_M_NR.

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 22254 AUXFU_ASSOC_M0_VALUE or 22256 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).

Reaction: 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 an M function in machine data 22254 AUXFU_ASSOC_M0_VALUE or 22256 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.

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

Parameters: %1 = channel number
%2 = machine data identifier
%3 = if necessary, 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.

Reaction: 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 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 = machine data identifier.
%4 = machine data identifier.

" Siemens AG 2007 All Rights Reserved

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

Reaction: Mode group not ready.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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 = machine data identifier
%3 = if necessary, 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.

Reaction: 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

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.

Reaction: 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.

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

" Siemens AG 2007 All Rights Reserved

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 machine data array 20060 AXCONF_GEOAX_NAME_TAB. The channel axis number is stored with the same index in the 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 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.

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

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.

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]).

Reaction: 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.
Set MD 30310 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.

Reaction: NC not ready.
Mode group not ready, also effective for single axes
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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 MD array 20070 AXCONF_MACHAX_USED).

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

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]).

Reaction: 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.
Set MD 30300 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 spindle speed limitation or for dry run feedrate).

Reaction: 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.

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

Parameters: %1 = Program position

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.

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

Remedy: Please inform the authorized personnel/service department.
Take greater care when optimizing the clock times NC 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 highest load on the control. For safety, a margin of 15 to 25% should be added to the times determined in this way.

Program Continuation: Switch control OFF - ON.

4260 Machine data %1 illegal

Parameters: %1 = String: MD identifier

Definitions: The selected cam pair was not activated by MD 10450 SW_CAM_ASSIGN_TAB or more than one cam pair has been selected.

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

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

" Siemens AG 2007 All Rights Reserved

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.

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. Correct machine data.
Active required inputs/outputs via MD:
• MD 10350 FASTIO_DIG_NUM_INPUTS
• MD 10360 FASTIO_DIG_NUM_OUTPUTS
• MD 10300 FASTIO_ANA_NUM_INPUTS
• MD 10310 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.

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. Correct machine data.

Program Continuation: Switch control OFF - ON.

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

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

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

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 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.

" Siemens AG 2007 All Rights Reserved

4282 Hardware of external NCK outputs assigned repeatedly

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

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. 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

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 hardware.

Program Continuation: Switch control OFF - ON.

4290 Sign of life monitoring: local P-bus not alive

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.

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 hardware.

Program Continuation: Switch control OFF - ON.

4291 Failure of module in local P-bus slot %1, error codes %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.

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 hardware.

" Siemens AG 2007 All Rights Reserved

Program Continuation: Switch control OFF - ON.

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

Parameters: %1 = string: MD identifier
%2 = axis name, spindle number

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

Reaction: Alarm display.

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

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

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

Parameters: %1 = string: MD identifier
%2 = index in MD array

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

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

Remedy: Correct the MD

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

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 machine data specified cannot be activated at the same time for one axis.

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

Remedy: Deactivate one of the functions.

Program Continuation: Switch control OFF - ON.

4340 Channel %1 block %2 invalid transformation type in transformation no. %3

Parameters: %1 = channel number
%2 = transformation number

Definitions: An invalid, i.e. undefined, number was entered in one of the machine data 24100 TRAFO_TYPE_1. This alarm occurs only if a certain type of transformation is not possible in the specified control type.

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

Remedy: Valid transformation type entered

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

" Siemens AG 2007 All Rights Reserved

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

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

Remedy: Enter valid machine data.

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

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 or the input axes 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 basic orientation (TRAFO5_BASE_ORIENT_*) is not defined (all three entries of the vector are 0)
- 4: The first and second axis are (virtually) parallel
- 5: On TRAFO_TYPE = 56 (rotatable tool and workpiece) there is no 4-axis transformation, i.e. 2 rotary axes must always be available. (See MD TRAFO_AXES_IN_X)
- 6: The third axis (TRAFO5_AXIS3_*) is not defined (all three entries of the vector are 0) (6-axis transformation)
- 7: The normal tool vector (TRAFO6_BASE_ORIENT_NORMAL_*) is not defined (all three entries of the vector are 0) (6-axis transformation)
- 8: The basic tool orientation (TRAFO5_BASE_ORIENT_*) and the normal tool vector (TRAFO6_BASE_ORIENT_NORMAL_*) are (virtually) parallel (6-axis transformation)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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 in or to activate the machine data with RESET or NEWCONFIG.

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

Remedy: Set valid machine data

" Siemens AG 2007 All Rights Reserved

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.

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

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_1/2 contains an invalid entry.
The following causes for the error are possible:
- The entry references a channel axis that does not exist.
- The entry is zero (no axis) but the transformation needs the relevant axis as a channel axis.

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

Remedy: Correct the entry in TRAFO_GEOAX_ASSIGN_TAB_1/2 or TRAFO_AXES_IN_1/2.

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 the machine data
%3 = transformation number

Definitions: Machine data TRAFO_AXIS_IN_1/2 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.

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

Remedy: Correct the entry in TRAFO_AXES_IN_1/2.

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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. Resetting of 32410 JOG_AND_POS_JERK_ENABLE or MD 35240 ACCEL_TYPE_DRIVE.

Program Continuation: Switch control OFF - ON.

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

Definitions: An MD has been altered that configures the buffered memory. If the NC 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, ...).

Reaction: Alarm display.

Remedy: If the control includes user data that has not yet been saved, then a data backup must be performed before the next NC 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 necessary.

4402 %1 causes a machine data reset

Parameters: %1 = machine data

Definitions: If this machine data is set, the current values of the machine data are overwritten with the pre-set values on next power up. Under certain circumstances, this can result in loss of data (also in the backed-up memory).

Reaction: 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 necessary.

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 MD 20110 RESET_MODE_MASK Bit4 and Bit5, the reset behavior of the 6th or 8th G group was determined. This setting is now made in MD 20152 GCODE_RESET_MODE.
In order to ensure compatible handling of "old" data backups, the "old" values are taken from MD 20110 RESET_MODE_MASK and entered in MD 20152 GCODE_RESET_MODE.

Reaction: Alarm display.

Remedy:

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

" Siemens AG 2007 All Rights Reserved

4503 In TO unit, %1 H number %2 assigned more than once. H number linked again.

Parameters: %1 = TO unit
%2 = H number

Definitions: This error can only occur when MD $MN_MM_EXTERN_CNC_SYSTEM= 1 or 2. The Power ON effective machine data bit 10890, $MN_EXTERN_TOOLPROG_MODE, bit 3 has been reset. On reconstructing data handling after Power ON, it has been found that different edges of the same TO unit have the same H number. They had been linked previously. They are linked again and MD bit $MN_EXTERN_TOOLPROG_MODE, bit 3 is set again.

Reaction: Alarm display.

Remedy: H numbers must be assigned only once in a TO unit. Then, machine data bit 10890, $MN_EXTERN_TOOLPROG_MODE, bit 3 can be set = 0 and a restart can be performed.

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

4610 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Definitions: The handwheel module for handwheel %1 requested through machine date $MN_HANDWHEEL_MODULE is not available for 840D systems. An 840D system is always regarded as a module. For this reason, handwheels directly linked with 840D systems $MN_HANDWHEEL_MODULE = 1 must always be set.

Reaction: Interface signals are set.
Alarm display.

Remedy: Set machine date $MN_HANDWHEEL_MODULE = 1 for the corresponding handwheel.

Program Continuation: Switch control OFF - ON.

4611 Invalid handwheel input for handwheel %1

Parameters: %1 = Handwheel input

Definitions: The handwheel input for handwheel %1 requested through machine date $MN_HANDWHEEL_INPUT is not available for 840D systems. With 840D systems, a maximum of 3 handwheels can be linked directly (1st and 2nd directly to 840D HW, 3rd handwheel through free encoder input).

Reaction: Interface signals are set.
Alarm display.

Remedy: Configure machine date $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel

Program Continuation: Switch control OFF - ON.

4620 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Definitions: The handwheel module for handwheel %1 requested through machine date $MN_HANDWHEEL_MODULE is not available for 802D systems. An 802D system is always regarded as a module. For this reason, $MN_HANDWHEEL_MODULE = 1 must always be set for handwheels directly linked with 802D systems.

Reaction: Interface signals are set.
Alarm display.

Remedy: Set machine date $MN_HANDWHEEL_MODULE = 1 for the corresponding handwheel.

Program Continuation: Switch control OFF - ON.

4621 Invalid handwheel input for handwheel %1

Parameters: %1 = Handwheel input

Definitions: The handwheel input for handwheel %1 requested through machine date $MN_HANDWHEEL_INPUT is not available for 802D systems. With 802D systems, a maximum of 2 handwheels can directly be linked.

Reaction: Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Configure machine date $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel

Program Continuation: Switch control OFF - ON.

4630 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Definitions: The reference in $MN_HANDWHEEL_MODULE for a corresponding entry in machine date array $MN_HANDWHEEL_LOGIC_ADDRESS[] necessary for the configuration of PROFIBUS handwheels is not available.

Reaction: Interface signals are set.
Alarm display.

Remedy: Configure the machine data $MN_HANDWHEEL_MODULE for the corresponding PROFIBUS handwheel so that there is a valid reference to an entry in the machine data array $MN_HANDWHEEL_LOGIC_ADDRESS[].

Program Continuation: Switch control OFF - ON.

4631 Invalid handwheel slot for handwheel %1

Parameters: %1 = Handwheel slot

Definitions: The handwheel slot for handwheel %1 requested through the machine date $MN_HANDWHEEL_INPUT is not available for PROFIBUS handwheels.

Reaction: Interface signals are set.
Alarm display.

Remedy: Configure machine date $MN_HANDWHEEL_INPUT to a valid handwheel slot for the corresponding PROFIBUS handwheel.

Program Continuation: Switch control OFF - ON.

4632 Logical PROFIBUS handwheel slot base address for handwheel %1 not found

Parameters: %1 = Handwheel number

Definitions: The logical base address of the PROFIBUS handwheel slot in machine date array $MN_HANDWHEEL_LOGIC_ADDRESS[] indexed in machine date $MN_HANDWHEEL_MODULE was not found.

Reaction: Interface signals are set.
Alarm display.

Remedy: Check if $MN_HANDWHEEL_MODULE of the corresponding handwheel is correct. Check if indexed logical base address of PROFIBUS handwheel slot in machine date array $MN_HANDWHEEL_LOGIC_ADDRESS[] is correct.

Program Continuation: Switch control OFF - ON.

5000 Communication job not executable %1

Parameters:

Definitions: The communication request (data exchange between NC and HMI,
for example: load an NC part program) cannot be executed due lack of memory. Cause: too many parallel communication requests

Reaction: Alarm display.

Remedy: No remedy possible – the operation that produced to alarm must be repeated. Clear the alarm display with Cancel.

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

" Siemens AG 2007 All Rights Reserved

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 is available as dynamic or static memory for the NC user (e.g. for: the number of tool offsets, the number of directories and files, etc.) and is therefore not sufficient.

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

Remedy: Redefine the NC memory structure!
A specific MD 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 MD has been chosen too large and therefore it is advisable to reduce the memory area by a certain proportion in several MDs.

Program Continuation: Teileprogramm neu starten.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 (module name)
%3 = internal error ID

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 numberExplanation
1No memory space available
2Maximum number of possible symbols exceeded
3Index 1 lies outside valid range of values
4Name in channel already exists
5Name in NC 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, modifications to machine data that are already correct produce errors in the user memory configuration.
The following block names (2nd parameter) are known in the NC (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, NC global
_N_NC_SEA- System-internal: Setting data, NC global
_N_NC_TEA- System-internal: Machine data, NC global
_N_NC_CEA- System-internal: 'cross error compensation'
_N_NC_PRO- System-internal: Protection zones, NC global
_N_NC_GD1- User: 1st GUD block defined by
_SGUD_DEF, NC global
_N_NC_GD2- User: 2nd GUD block defined by
_N_MGUD_DEF, NC global
_N_NC_GD3- User: 3rd GUD block defined by
_N_UGUD_DEF, NC global
_N_NC_GD4- User: 4th GUD block defined by
_N_GUD4_DEF, NC global
_N_NC_GD5- User: 5th GUD block defined by
_N_GUD5_DEF, NC global
_N_NC_GD6- User: 6th GUD block defined by

" Siemens AG 2007 All Rights Reserved

_N_GUD6_DEF, NC global
_N_NC_GD7- User: 7th GUD block defined by
_N_GUD7_DEF, NC global
_N_NC_GD8- User: 8th GUD block defined by
_N_GUD8_DEF, NC global
_N_NC_GD9- User: 9th GUD block defined by
_N_GUD9_DEF, NC global
_N_NC_MAC- User: Macro definitions
_N_NC_FUN- User: Cycle programs
_N_CHc_OPT- System-internal: Option data, channel-specific
_N_CHc_SEA- System-internal: Setting data, channel-specific
_N_CHc_SEA- 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 parameters, channel-specific
_N_CHc_GD1- User: 1st GUD block defined by
_N_SGUD_DEF, channel-specific
_N_CHc_GD2- User: 2nd GUD block defined by
_N_MGUD_DEF, channel-specific
_N_CHc_GD3- User: 3rd GUD block defined by
_N_UGUD_DEF, channel-specific
_N_CHc_GD4- User: 4th GUD block defined by
_N_GUD4_DEF, channel-specific
_N_CHc_GD5- User: 5th GUD block defined by
_N_GUD5_DEF, channel-specific
_N_CHc_GD6- User: 6th GUD block defined by
_N_GUD6_DEF, channel-specific
_N_CHc_GD7- User: 7th GUD block defined by
_N_GUD7_DEF, channel-specific
_N_CHc_GD8- User: 8th GUD block defined by
_N_GUD8_DEF, channel-specific
_N_CHc_GD9- User: 9th 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: tool holder 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

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

Remedy: Correct the machine data or undo the change.
Please inform the authorized personnel/service department.
There are two determining machine data for cycle programs:
MD 18170 MM_NUM_MAX_FUNC_NAMES = max. number of all
Cycle programs

" Siemens AG 2007 All Rights Reserved

Error number = 2 shows that this value is too small
MD 18180 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 this MD is modified, the memory backup is retained)
The following applies to macro definitions:
MD 18160 MM_NUM_USER_MACROS = max. number of all Macro definitions
Error number = 2 shows that this value is too small
(If this MD is modified, the memory backup is retained)
The following applies to GUD variables:
MD 18118 MM_NUM_GUD_MODULES = max. number of all GUD data blocks per area (NC/channel)
(if GD1, GD2, GD3, GD9 are defined, the value must be 9
MD 4 MM_NUM_ GUD_NAMES_NCK = max. number of all global GUD variables
Error number = 2 shows that this value is too small
MD 18130 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
MD 18150 MM_GUD_VALUES_MEM = max. memory values all GUD variables together
Error number = 1 shows that this value is too small
Switch the control OFF - ON.

Program Continuation: Switch control OFF - ON.

6020 Machine data have been changed - 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.

Reaction: Alarm display.

Remedy: No remedial action is 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.

Reaction: Alarm display.

Remedy: No remedial action is 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 defined for the control model in KB
%2 = actual maximum amount of free memory in KB
%3 = type of memory, "D" = non-buffered, "S" = buffered

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 18050 INFO_FREE_MEM_DYNAMIC, 18060 INFO_FREE_MEMS_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.

Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: The cause of this message may be that the NCK is running on hardware that is not intended for this NCK version (i.e. it has insufficient memory).
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.

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

6200 Memory for CC MD full.

Definitions: The memory reserved for storage of compile cycle machine data is full.
Some of these machine data could not be created correctly.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
If the alarm is displayed on start-up of compile cycles, this may be remedied by increasing $MN_MM_CC_MD_MEM_SIZE.

Program Continuation: Switch control OFF - ON.

6410 TO unit %1 tool %2 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- or quantity-monitored tool.
If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
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 is specified for this reason.

Reaction: Interface signals are set.
Alarm display.

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 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- or quantity-monitored tool.
If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
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.

Reaction: Interface signals are set.
Alarm display.

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

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

" Siemens AG 2007 All Rights Reserved

6412 TO unit %1 tool %2 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 monitoring limit for a time- or quantity-monitored tool.
If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
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 is specified for this reason.

Reaction: Interface signals are set.
Alarm display.

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 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 monitoring limit for a time- or quantity-monitored tool.
If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
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.

Reaction: Interface signals are set.
Alarm display.

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

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

6415 TO unit %1 tool %2 with tool edge no. %3 has reached tool monitor warning limit

Parameters: %1 = TO unit
%2 = Tool identifier
%3 = Cutting edge number

Definitions: This message informs that at least one cutting edge of the time or quantity monitored tool has reached its monitoring limit. The alarm was triggered through the OPI interface (MMC, PLC). The channel context is not defined. The TO unit was specified for this reason.

Reaction: Interface signals are set.
Alarm display.

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

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

6416 Channel %1 tool %2 with tool edge no. %3 has reached tool monitor warning limit

Parameters: %1 = Channel number
%2 = Tool identifier
%3 = Cutting edge number

" Siemens AG 2007 All Rights Reserved

Definitions: This message informs that at least one cutting edge of the time or quantity monitored tool has reached its monitoring limit. The limit was detected in the channel context. The alarm originated during NC program execution.

Reaction: Interface signals are set.
Alarm display.

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

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

6417 TO unit %1 tool %2 with tool edge no. %3 has reached tool monitoring limit

Parameters: %1 = TO unit
%2 = Tool identifier
%3 = Cutting edge number

Definitions: This message informs that at least one cutting edge of the time or quantity monitored tool has reached its monitoring limit. The alarm was triggered through the OPI interface (MMC, PLC). The channel context is not defined. The TO unit was specified for this reason.

Reaction: Interface signals are set.
Alarm display.

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

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

6418 Channel %1 tool %2 with tool edge no. %3 has reached tool monitoring limit

Parameters: %1 = Channel number
%2 = Tool identifier
%3 = Tool number

Definitions: This message informs that at least one cutting edge of the time or quantity monitored tool has reached its monitoring limit. The limit was detected in the channel context. The alarm originated during NC-program execution.

Reaction: Interface signals are set.
Alarm display.

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

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

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

Definitions: No further entry of cutting edges possible in the workpiece 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 is used precisely once for a workpiece then the limit is reached.
If several workpieces are made on several tool holders/spindles simultaneously, it is possible to note MD18100 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).

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

Remedy: - Have you forgotten to decrement the workpiece counter?
Then program SETPIECE in the part program or in the PLCprogram, insert the command correctly.
- If the part program or the PLC is correct, then more memory should be set for the tool cutting edges via machine data 18100 MM_NUM_CUTTING_EDGES_IN_TOA (only people with proper access rights can do this!).

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

" Siemens AG 2007 All Rights Reserved

6431 Channel %1 block %2 Function not allowed. Tool management/monitoring is not active.

Parameters:

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start disable.
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? And now this part program is started on the numerical control without tool management/tool monitoring This does not work. 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 MD 18080 MM_TOOL_MANAGEMENT_MASK,
MD 20310 TOOL_MANAGEMENT_MASK.
Check that the necessary option is set.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6452 Channel %1 block %2 tool change not possible. Tool holder/spindle number = %3 not defined.

Parameters: %1 = Channel ID
%2 = Block number, label
%3 = Tool holder/spindle number

Definitions: The desired tool change is not possible. The toolholder/spindle number has not been defined.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: General: The following must apply: 'maximum programmed address extension s (=spindle number/toolholder number)
of Ts=t, Ms=6 must be less than the value of $MN_MM_NUM_LOCS_WITH_DISTANCE.
With magazine management: Check whether the toolholder number/spindle number and the magazine data have been defined correctly.
(See also the system variables $TC_MPP1, $TC_MPP5 of the buffer magazine).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

6500 NC memory full

Definitions: Too many part programs have been unloaded. The task cannot be executed.
During first start-up the files in the NC file system (part of the NC memory) may be affected, for example initialization files, NC programs, etc.

Reaction: Alarm display.

Remedy: Delete or empty files (e.g. part programs).

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

6510 Too many part programs in the NC memory

Definitions: The number of files in the NC file system (part of the NC memory) has reached the maximum number possible.

Reaction: Alarm display.

Remedy: Delete or empty files (e.g. part programs).

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

" Siemens AG 2007 All Rights Reserved

6530 Too many files in directory

Definitions: The number of files in one directory of the NC has reached the maximum limit.
Reaction: Alarm display.
Remedy: Delete or empty files (e.g. part programs) in the subdirectory affected.
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 NC file system (part of the NC memory) has reached the maximum number possible.
Reaction: Alarm display.
Remedy: Delete or empty directories (e.g. workpiece) that are not required.
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 NC has reached the maximum limit.
Reaction: Alarm display.
Remedy: Please inform the authorized personnel/service department.
Delete or empty subdirectories in the respective directory, or
Program Continuation: Clear alarm with the Delete key or NC START.

6560 Data format not allowed

Definitions: Illegal data have been entered in an NC file, e.g. binary data loaded as an ASCII file.
Reaction: Alarm display.
Remedy: Mark the file as binary data (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 NC is full. The task cannot be executed. Too many system files were created in the DRAM.
Reaction: 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
Reaction: Alarm display.
Remedy: Delete or empty files (e.g. part programs).
Program Continuation: Clear alarm with the Delete key or NC START.

6581 NC user memory full

Definitions: The DRAM file system of the user area is full. The order cannot be executed.
Reaction: Alarm display.
Remedy: Delete or unload files (e.g. parts programs)
Program Continuation: Clear alarm with the Delete key or NC START.

6582 NC machine OEM memory full

Definitions: The DRAM file system of the machine OEM area is full. The order cannot be executed.
Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Delete or unload files (e.g. parts programs)
Program Continuation: Clear alarm with the Delete key or NC START.

6583 NC system memory full

Definitions: The DRAM file system of the system area (Siemens) is full. The order cannot be executed.
Reaction: Alarm display.
Remedy: Delete or unload files (e.g. parts programs)
Program Continuation: Clear alarm with the Delete key or NC START.

6584 NC memory limit TMP reached

Definitions: The DRAM file system of the TMP (temporary) area is full. The job cannot be executed.
Reaction: Alarm display.
Remedy: Increase machine date $MD_MM_DRAM_FILE_MEM_SIZE or switch off the precompilation of individual or all cycles or, if need be,
delete the files in the TMP area.
Program Continuation: Clear alarm with the Delete key or NC START.

6585 NC external memory limit reached

Definitions: The DRAM file system of the external area (execution of the external drive) is full.
The job cannot be executed.
Reaction: Alarm display.
Remedy: Load the files to be executed explicitly into the NCK.
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 NC is full. No more data can be stored on the NC card.
Reaction: 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 accessed simultaneously on the NC card
Reaction: 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.
Reaction: 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.
Reaction: Alarm display.
Remedy: Replace the PCMCIA card.
Program Continuation: Clear alarm with the Delete key or NC START.

" Siemens AG 2007 All Rights Reserved

6640 NC card is not inserted

Definitions: The NC card cannot be accessed because the card is not plugged in.

Reaction: 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.

Reaction: 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.

Reaction: 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.

Reaction: Alarm display.

Remedy: Wait until the read-out procedure is terminated

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

6671 NC card write active

Definitions: The alarm is active while the contents of the NC card are being written.
The Flash File System cannot be accessed during this period.
If a Power Off occurs while the alarm is present, the contents of the NC will be destroyed!

Reaction: Alarm display.

Remedy: Wait until the write procedure is terminated.

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

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_FUNCTION_MASK can be copied from the NC card to the passive file system.

Reaction: 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_FUNCTION_MASK can be saved. It is possible that cycles are lost during the next booting.

Reaction: 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 2007 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.

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

Remedy: Import the cycle again.

Program Continuation: Switch control OFF - ON.

6693 File %1 lost

Parameters: %1 = file name

Definitions: Due to a power failure, a file change could not be terminated properly. The file is lost.

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

Remedy: Import the file again.

Program Continuation: Switch control OFF - ON.

6698 Unknown NC card (%1/%2). Writing not possible.

Parameters:

Definitions: The NC card cannot be accessed because a valid write algorithm is not available for the flash memory.

Reaction: Alarm display.

Remedy: Use a compatible NC card or enter the new manufacturer code/device code in MD 11700 PERMISSIVE_FLASH_TAB after consultation with SIEMENS.

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

7500 Block %1 invalid protection level for command %2 (protection level act.: %3 prog.: %4)

Parameters: %1 = block number
%2 = programmed command
%3 = current protection level of command
%4 = programmed protection level of command

Definitions: During assignment of a protection level to a part program command using the REDEF command,
an non-permitted part program command was programmed
a protection level was programmed with a logically smaller number
(higher value) that the protection level that currently applies to this
command.
the relevant definition file does not possess sufficient write protection. The write protection for the file must be at least as high as the highest protection level that is assigned to part program command in his definition file.

Reaction: Alarm display.

Remedy: Correct the definition files /_N_DEF_DIR/_N_MACCESS_DEF or /_N_DEF_DIR/_N_UACCESS_DEF. The language commands that are permissible for the relevant system configuration are given in the Siemens Programming Guide or in the Manufacturer's documentation.

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

" Siemens AG 2007 All Rights Reserved

8000 Channel %1 option 'user interrupt programs' not set

Parameters: %1 = channel number

Definitions: The signals of NCK inputs are required in order to activate the interrupt routines and rapid lift from contour. This function is not included.

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

Remedy: Do not use fast interrupt inputs!

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 MD 20070 AXCONF_MACHAX_USED than are allowed in the system.

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

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.

Program Continuation: Switch control OFF - ON.

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.

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

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.

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

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.

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

Remedy: Please inform the authorized personnel/service department.
- Buy option
- Activate less SRAM

" Siemens AG 2007 All Rights Reserved

Program Continuation: Switch control OFF - ON.

8023 Option 'Activation of more than %1 KB PLC user memory' not set

Parameters: %1 = Memory size

Definitions: The option for the memory configuration does not correspond to the PLC user memory used.

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

Remedy: Please inform the authorized personnel/service department.
- Purchase option
- Use less PLC user memory

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.

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

Remedy: Only so many axes can be programmed in the part program as correspond 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.

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

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.

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

Remedy: - Buy option
- Do not configure any containers

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

" Siemens AG 2007 All Rights Reserved

8037 Option 'APC activation' not set

Definitions: The function 'Advanced Positioning Control' (APC) was activated in the control although the relevant option has been set.

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

Remedy: Deactivate the function 'Advanced Positioning Control' (APC) in the drive.

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.

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

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 via an option.

Reaction: Alarm display.

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. The functions that belong to the option have been selected for too many axes in the specified axial machine data.

Reaction: 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:

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

8044 Option for IPO cycle time %1 ms not set

Parameters: %1 = illegal IPO cycle

Definitions: The option for activation of an IPO cycle time of %1 ms has not been set.
Option - Permiss. IPO cycle time:
• Option-free >= 8 ms
• 1st step >= 6 ms
• 2nd step >= 4 ms
• 3rd step >= 2 ms
• 4th step < 2 ms

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

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.

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

Remedy: - Buy option
- Set (current/speed controller) cycle times to 810D default values.

Program Continuation: Switch control OFF - ON.

8050 Option 'SPL inputs/outputs' not set.

Definitions: The number of PLC I/Os has not been set in the option date for Solution Line.

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

Remedy: Select area SI Basic or SI Comfort in the option date.

Program Continuation: Switch control OFF - ON.

8051 Option 'Handwheel on PROFIBUS' not set

Definitions: The option to operate handwheels on PROFIBUS is not set.

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

Remedy: Activate option 'Handwheel on PROFIBUS'

Program Continuation: Switch control OFF - ON.

8080 %1 option(s) is/are activated without setting the license key

Parameters: %1 = Number of non-licensed options

Definitions: Options were activated but no license key set to prove that they were purchased.

Reaction: Alarm display.

Remedy: Generate license key via Internet and enter.
Take back option.

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

" Siemens AG 2007 All Rights Reserved

8081 %1 option(s) is/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.

Reaction: Alarm display.

Remedy: Generate license key via Internet and enter.

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

8082 A wrong license key was entered three times, Power On required before next try.

Definitions: The license can be entered three times max (correctly or incorrectly).

Reaction: Alarm display.

Remedy: Execute NCK Power On and enter the license key (correctly).
Take back option.

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

8088 'Selection of non-grinding-specific tools' option not possible

Definitions: The system version of the software only allows selection of grinding specific tools (i.e. tools of type 4xx).

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

Remedy: Select a tool of type 4xx (grinding tool)
or install a standard version of the system software

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

8098 Invalid combination of options (%1)

Parameters: %1 = Bit mask of options

Definitions: The following restrictions apply to this module for the combination of options:
The option "Two-channel" and the options "External language", "Nibbling", "Neural quadrant error compensation" and "Measurement level 2" exclude one another!
Bit0 (LSB): Nibbling
Bit1 : External language
Bit2 : Neural quadrant error compensation
Bit3 : Measurement level 2

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

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. Synchronized 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 2007 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...").

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

Remedy: Modify part program.

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

8120 Channel %1 block %2 following axis/spindle %3 generic coupling %4 required

Parameters: %1 = Channel number
%2 = Block number
%3 = Slave axis
%4 = String

Definitions: The option stage is inadequate for the desired function. Possible reasons:
More couplings have been created than are permitted.
The number of permissible leading axes has been exceeded for one or more couplings.
The range of functions of one or more couplings has not been released.

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

Remedy: Buy an adequate option stage.
Reduce the number of simultaneously active couplings.
Reduce the number of leading axes per coupling or only use the released range of functions.

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

9000 Handwheel %1 failed

Parameters: %1 = Handwheel number

Definitions: PROFIBUS handwheel failed

Reaction: Interface signals are set.
Alarm display.

Remedy: Restore connection to PROFIBUS handwheel

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

10203 Channel %1 NC start without reference point (action=%2<ALNX>)

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.

Reaction: Interface signals are set.
Alarm display.

Remedy: The start of referencing can be enabled channel-specific or axis-specific.
1.Channel-specific reference point approach: The rising edge of the interface signal "activate referencing" (V 32000001.0) starts an automatic sequence which starts the axes of the channel in the same sequence as specified in the axis-specific MD 34 110 REFP_CYCLE_NR (axis sequence channel-specific referencing).
-1:The axis does not participate in channel-specific referencing but must be referenced for the NC Start.
0:The axis does not participate in channel-specific referencing but does not have to be referenced for the NC Start.
1-4:Starting sequence for the channel-specific referencing (simultaneous start with same number).
2.Axis-specific referencing: Press direction button that corresponds to the direction of approach in the axis-specific MD 34 010 REFP_CAM_DIR_IS_MINUS (approach reference point in minus direction).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

Reaction: Interpreter stop
Alarm display.
NC Stop on alarm.

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 only initiates the alarm response NC Stop.
The alarm is output in the following situations:
- If $MN_SEARCH_RUN_MODE bit 0 ==1 and the last action block is loaded in the main run after block search. Alarm 10208 is then activated as a function of the VDI signal PLC -> NCK channel DBB1.6.
- Search alarm 10208 has been suppressed by the PI service _N_FINDBL (third decade of the parameter supplied with "2"). Alarm 10209 is set as a function of whether or not a search ASUB has been configured ($MN_SEARCHRUN_MODE bit 1) with the end of the search ASUB or the loading of the last action block in the main run.

Reaction: Interpreter stop
NC Stop on alarm.

Remedy: NC-Start

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10225 Channel %1: command denied

Parameters: %1 = channel number
%2 = string (event name)

Definitions: The channel contains a command that cannot be executed.

Reaction: Alarm display.

Remedy: Press RESET

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

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

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

" Siemens AG 2007 All Rights Reserved

10261 Channel %1 communication overload for block preparation

Parameters: %1 = Channel number

Definitions: The internal communication between the NCK modules that evaluate the channel-specific VDI signals (START/STOP/RESET/DDTG/ASUBS/...) and the block preparation are overloaded. The block preparation modules are not being allocated enough computing time.

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

Remedy: More processor time must be allocated to the block preparation modules. Machine data $MN_IPO_SYSCLOCK_TIME_RATIO or $MN_SYSCLOCK_CYCLE_TIME can be increased for this purpose.

Program Continuation: Switch control OFF - ON.

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.

Reaction: Alarm display.

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.

Reaction: 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 remedies:
• 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. Possible causes could be:
G09
Auxiliary function after motion
Auxiliary function output before the motion of the following block

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

Remedy: Modify the NC part program by removing any programmed "Stop at end of block" G09.
Modify general MD 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 the motion
Bit 6 = 1:Auxiliary function output during the motion
Bit 7 = 1:Auxiliary function output after the motion

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

" Siemens AG 2007 All Rights Reserved

10602 Channel %1 block %2 velocity limitation during thread cutting

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

Definitions: In the displayed thread block, the axis would exceed its maximum velocity when the spindle override is in the maximum position.

Reaction: Local alarm reaction.
Alarm display.

Remedy: If the axis velocity is not limited (faultless thread) no remedial measures are necessary. Otherwise, a lower spindle speed must be programmed for the thread block.

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

10604 Channel %1 block %2 thread lead increase too high

Parameters: %1 = channel number
%2 = block number, label

Definitions: The thread lead increase cause axis overload.
A spindle override of 100% is assumed during verification.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Reduce the spindle speed, thread lead increase or path length in the part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Reduce the thread lead decrease or path length in the part 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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Perform thread cutting with G33, G34, G35 without 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

" Siemens AG 2007 All Rights Reserved

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!

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

Remedy: Check and correct the part program (analyze whether motion beyond block boundaries is appropriate here). Prevent block change by means of the vocabulary word 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

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. Unable to detect overstepping of the traversing area because a motion overlay was made by the handwheel.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: The following remedies apply, depending on the cause:
Handwheel override Cancel the motion overlay and avoid this or keep it smaller when the program is repeated.
Check the transformation of 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.

Reaction: Alarm display.

Remedy: The machine data:
MD 36110 POS_LIMIT_PLUS/MD 36130 POS_LIMIT_PLUS2 and MD 36100 POS_LIMIT_MINUS/MD 36120 POS_LIMIT_MINUS2 must be checked for the software limit switches.
Check the axis-specific interface signals: "2. Check whether 2nd software limit switch is selected with 2nd software limit switch "Plus" (V 380x1000.3) and 2nd software limit switch "Minus" (V 380x1000.2).

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Program a different motion or do not perform a motion overlay.

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 positive working area limitation in JOG mode.

Reaction: Alarm display.

Remedy: Setting data:
Check SD 43420 WORKAREA_LIMIT_PLUS and
SD 43430 WORKAREA_LIMIT_MINUS for work area limitation.

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

10632 Channel %1 block %2 axis %3 reaches the coordinate system-specific 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 coordinate system-specific working area limitation. This is not detected until the main run, either because the minimum axis values could not be determined before the transformation or because there is an overlaid movement.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Program other motion or do not perform overlaid motion.

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

10633 Channel %1 axis %2 is at coordinate system-specific working area limit %3

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = String (+ or -)

Definitions: The specified axis reaches the coordinate system-specific working area limitation in JOG mode.

Reaction: Alarm display.

Remedy: Check the system parameter $P_WORKAREA_CS_xx for the coordinate system-specific working area limitation.

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

" Siemens AG 2007 All Rights Reserved

10700 Channel %1 block %2 NCK protection zone %3 violated during automatic or MDI 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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.
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 MDI 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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.
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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Protection zone can be traversed after a new NC Start.

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

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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Protection zone can be traversed after a new NC Start.

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

" Siemens AG 2007 All Rights Reserved

10704 Channel %1 block %2 protection zone monitoring is not guaranteed

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

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.

Reaction: Interface signals are set.
Alarm display.

Remedy: Take other measures to ensure that the geomotry 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 necessary.

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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

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 necessary.

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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

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 necessary.

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.

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

Remedy: Modify program.

" Siemens AG 2007 All Rights Reserved

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 2nd software limit switch is selected with interface signal "2nd software limit switch Plus/Minus" in V 380x1000.2 and 0.3). The alarm is activated when preparing the part program block.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Check the position of the axis as specified in the part program.
Check MD 36110 POS_LIMIT_MINUS/MD 36130 POS_LIMIT_MINUS2 and MD 36100 POS_LIMIT_PLUS/MD 36120 POS_LIMIT_PLUS2 for the software limit switches.
Check the axis-specific interface signals: "2. Check whether 2nd software limit switch is selected with software limit switch "Plus/Minus" (V 380x1000.2 and .3).
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: The planned motion violates the software limit switch for the axis. The alarm is activated during the preparation of approach or rest blocks at REPOS.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Check in the NC program and current positions. Check whether 2nd software limit switch is selected with axis-specific interface signal "2nd software limit switch Plus/Minus" (V 380x1000.2 and .3).
Check currently active zero offsets 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

10722 Channel %1 block %5 axis %2 software limit switch %6 violated, residual distance: %7 %3<ALUN> violated

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|number+string(+/-)|residual distance

Definitions: The path programmed for the axis violates the currently valid software limit switch. The alarm is activated when preparing the part program block.
This alarm is issued instead of alarm 10720 if bit 11=1 in the machine data $MN_ENABLE_ALARM_MASK. Alarm 10722 offers an expanded diagnostics option for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
See also diagnostics guide for alarm 10720.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Check and correct positions in the NC program.
Please inform the authorized personnel/service department.
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 work offsets via the current frame.
Work offsets, overlaid movements ($AA_OFF), DRF and transformations components must also be checked.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10723 Channel %1 block %5 axis %2 software limit switch %6 violated, residual distance: %7 %3<ALUN>

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|number+string(+/-)|residual distance

Definitions: The motion planned for the axis violates the currently active software limit switch.
The alarm is activated during the preparation of approach or rest blocks for REPOS.
This alarm is issued instead of alarm 10721 if bit11=1 in machine data $MN_ENABLE_ALARM_MASK. Alarm 10723 offers an expanded diagnostics option for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
See also the Diagnostics Guide for alarm 10721.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs.
Please inform the authorized Personnel/Service.
Check the machine data: 36100 POS_LIMIT_MINUS / 36120 POS_LIMIT_MINUS2 and 36110 POS_LIMIT_PLUS / 36130 POS_LIMIT_PLUS2 for the software limit switch.
Check the axis-specific interface signals "2nd software limit switch plus/minus" (DB31-61, DBX 12.2 and 12.3) to see whether the 2nd software limit switch is selected.
Check the currently active work offset via the current frame.
Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components.
Cancel the NC program with NC reset.

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

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: a)Check the NC program for correct position data.
b)Check zero offsets (current frame)
c)Correct work area limitation with G25 or
d)Correct work area limitation via setting data or
Deactivate working area limitation via SD 43410 WORKAREA_MINUS_ENABLE=FALSE

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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

10732 Channel %1 block %5 axis %2 working area limitation violated, residual distance: %6 %3<ALUN>

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|residual distance

Definitions: This alarm is generated if it is determined during block preparation that the programmed path of the stated axis violates the working area limitation.
This alarm is issued instead of alarm 10730 if bit 11=1 in machine data $MN_ENABLE_ALARM_MASK. Alarm 10732 offers an expanded diagnostics option for the working area limitation violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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/G26, 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.

10733 Channel %1 block %5 axis %2 working area limitation violated, residual distance: %6 %3<ALUN>

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|residual distance

Definitions: The motion planned for the axis violates the currently active working area limitation.
The alarm is activated during the preparation of approach or rest blocks for REPOS.
This alarm is issued instead of alarm 10731 if bit11=1 in machine data $MN_ENABLE_ALARM_MASK. Alarm 10733 offers an expanded diagnostics option for the working area limitation violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs.
Check the currently active work offset via the current frame.
Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components.
Cancel NC program with NC reset.

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

" Siemens AG 2007 All Rights Reserved

10735 Channel %1 block %5 axis %2 coordinate system-specific working area limitation violated, residual distance: %6 %3<ALUN>

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|residual distance

Definitions: This alarm is generated if it is determined during block preparation that the programmed path of the stated axis violates the coordinate system-specific working area limitation.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: a) Check NC program for correct positional data and, if necessary, make corrections.
b) Check work offsets (current frame)
c) Correct the working area limitation with WALCS1 ... WALCS9, or
d) Correct the working area limitation in $P_WORKAREA_CS_LIMIT_PLUS or $P_WORKAREA_CS_LIMIT_MINUS, or
e) Deactivate the working area limitation with $P_WORKAREA_CS_MINUS_ENABLE =FALSE or $P_WORKAREA_CS_PLUS_ENABLE.
In cases d) and e), then reactivate the group of the selected coordinate system-specific working area limitation.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10736 Channel %1 block %5 axis %2 coordinate system-specific working area limitation violated, residual distance: %6 %3<ALUN>

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Unit of distance
%4 = Block number, label|residual distance

Definitions: This alarm is generated if it is determined during block preparation that the programmed path of the stated axis violates the coordinate system-specific working area limitation.
The alarm is activated during the preparation of approach or residual blocks for REPOS.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs.
Check the currently active work offset via the current frame.
Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components.
Cancel NC program with NC 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: No more than 5 blocks may be programmed between the SAR block and the block which defines the approach or retraction tangent.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

10741 Channel %1 block %2 direction reversal with WAB infeed motion

Parameters: %1 = channel number
%2 = block number, label

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 SAR contour.

Reaction: 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.

10742 Channel %1 block %2 WAB distance invalid or not programmed

Parameters: %1 = channel number
%2 = block number, label

Definitions: Possible causes:
In a SAR 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 SAR contour is negative. The internally generated SAR 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.

Reaction: 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.

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 SAR motion before a previously activated SAR motion was terminated.

Reaction: 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.

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 further 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 SAR motion is vertical to the current machining plane.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Modify part program.

" Siemens AG 2007 All Rights Reserved

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 SAR block and in the following block, the position has been programmed perpendicular to the machining direction. In the SAR block, no position has been indicated in the machining plane.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Modify part program.
Either remove the position data for the infeed axis from the SAR block or the following block, or program a position in the machining plane in the SAR 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 SAR approach block and the following block defining the tangent direction or between a SAR approach block and the following block defining the end position.

Reaction: 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.

10747 Channel %1 block %2 retraction direction not defined for WAB

Parameters: %1 = channel number
%2 = block number, label

Definitions: In a SAR 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Modify part program.
The following modifications are possible:
Indicate end point in the machining plane in the SAR 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.

" Siemens AG 2007 All Rights Reserved

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 SAR movement.

Reaction: 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 tool radius compensation (TRC) 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.

Reaction: Correction block is reorganized.
Interpreter stop
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Before calling the TRC 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. So it may be that one of the equidistant paths violates the workpiece contour.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: 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, since the equidistants are extended or added to always obtain an intersection point).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

10752 Channel %1 block %2 overflow of local block buffer with tool radius compensation

Parameters: %1 = channel number
%2 = block number, label

Definitions: The tool radius compensation must buffer a varying number of intermediate blocks, in order to calculate 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 and the number of contour elements added.
The size of the buffer is fixed by the system and cannot be changed via the MDs.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: 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).

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 tool 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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.

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 tool 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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 options are available:
Select in the previous block
Add intermediate block
Select approach behavior NORM

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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 options are available:
Deselection in the next block
Add intermediate block
Select retraction behavior NORM

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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.
If the compensation value varies its sign within a block, both sides of the contour are checked, otherwise only the compensation side.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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 groove).

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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. 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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 number of empty blocks is limited.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: 1.Modify part program:
2.Check whether SBL2 has been selected. 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.

Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: The behavior is correct at narrow locations that cannot be machined with the active tool.
Modify part program.
Use tool with smaller radius if necessary.

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.

Reaction: 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.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: 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.

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

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.

Reaction: 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.

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

" Siemens AG 2007 All Rights Reserved

Definitions: This alarm occurs when an axis that is required for tool radius compensation is not a geometry axis.

Reaction: 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.

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 suppression of compensation for tool radius compensation is limited.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Modify part program.
Check whether SBL2 has been selected. 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), 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.

Reaction: Alarm display.
NC Stop on alarm at block end.

Remedy: Continue machining with CANCEL and Start.
Modify part program.

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

10780 Channel %1 block %2 impermissible change of a turning or grinding tool 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), 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Continue machining with CANCEL and Start.
Modify part program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

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.

Reaction: 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.

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

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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.

" Siemens AG 2007 All Rights Reserved

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 Bevel cutter
• 156 Bevel cutter
• 157 Bevel cutter

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Use a different 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.

Reaction: 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.

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.

Reaction: 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.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Modify part program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

10794 Channel %1 block %2 angle specification missing in 2nd 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.

Reaction: 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.

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.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

10800 Channel %1 block %3 axis %2 is not a geometry axis

Parameters: %1 = channel number
%2 = axis name, label
%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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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.

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 vocabulary word SETMS in the part program, thus allowing each spindle of the channel to be redefined as master spindle.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Pre-set master spindle with MD 20090 SPIND_DEF_MASTER_SPIND[n]=m (n ... channel index, m ... spindle No.) or define with an identifier in the NC part program before a G function that requires a master spindle is programmed.
The machine axis to drive the spindle must have a spindle number assigned in
MD 35000 SPIND_ASSIGN_TO_MACHAX[n]=m (n ... machine axis index,
m ... spindle No.). Furthermore, it must be assigned to a channel (channel index 1 or 2) via MD 20070 AXCONF_MACHAX_USED[n]=m (n ... channel axis index, m ... machine axis index).

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Correct part program or set the SD 43300 ASSIGN_FEED_PER_REV_SOURCE correctly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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).

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Change the program so it cannot reference itself.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

10870 Channel %1 block %2 facing axis for constant velocity not defined

Parameters: %1 = channel number
%2 = block number, label

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 MD 20100 DIAMETER_AX_DEF[n,m]=x (n ... channel index, m ... spindle index, x ... axis name), the name of the face axis [String] can be specified for each of the 5 spindles and it is used for speed calculation.
S [1/min] = "+++OLE"

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Enter the name of the facing axis in MD 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 the contour elements and to be joined with a chamfer or a radius (CHF, RND), too many dummy blocks have been programmed without contour information.
Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
Remedy: Modify the part program so that the maximum permissible number of empty blocks is not exceeded.
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.
Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
Remedy: Modify the part program so that number of empty 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
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 maximum permissible number of empty blocks with traversing information (dummy blocks) has been exceeded.
Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
Remedy: Correct the part program according to the error described above.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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 MD 11411 ENABLE_ALARM_MASK. Otherwise, the chamfer or radius is adapted without an alarm being output.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.
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.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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 irregular velocity waveform 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.

Reaction: Local alarm reaction.
Alarm display.

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.

Reaction: Interpreter stop
Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

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

" Siemens AG 2007 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 specified curve cannot be pre-calculated reliably. 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.

Reaction: 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 LookAhead 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.

Reaction: 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 causes could be:
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..).
- Incorporation of the tool length compensation depends on the selected working plane (see G17,..).
The machine stops before the faulty block.

Reaction: Interpreter stop
Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Modify the part program.
Modify an incorrectly specified tool length compensation.

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

" Siemens AG 2007 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 contour of the stock removal cycle (CYCLE 95) contains different path commands than: G00, G01, G02, G03, CIP or CT. The contour program may only contain contour elements that derive from these path conditions (i.e. no thread blocks, no spline blocks, etc.).

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Only program path elements that consist of straight lines or circles as a stock removal contour.

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 contour of the stock removal cycle (CYCLE 95) contains different path commands than:
Full circle
Overlapping contour elements
Wrong start position

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The above errors must be corrected in the program 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 stock removal cycle CYCLE 95 was interrupted during a preparatory phase of the stock removal contour.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: No interruptions are permitted during contour preparation in the stock removal cycle CYCLE 95.

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

10933 Channel %1 block %2 contour programm does not contain enough contour blocks

Parameters: %1 = channel number
%2 = block number, label

Definitions: The program in which the stock removal contour is programmed contains fewer than 3 blocks with motions in both axes in the machining plane. The cutting cycle (CYCLE 95) has been aborted.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Increase the size of the program 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

" Siemens AG 2007 All Rights Reserved

10934 Channel %1 block %2 array for contour segmentation is set too small

Parameters: %1 = channel number
%2 = block number, label

Definitions: The program in which the stock removal contour is programmed contains too many blocks with motions in both axes in the machining plane (CYCLE 95).

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The number of blocks in the contour program must be reduced. You should check the distribution of the contour over several programs.

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.

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

Remedy: Correct the part program and start it again.

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

Reaction: 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.

12000 Channel %1 block %2 address %3 programmed repeatedly

Parameters: %1 = channel number
%2 = block number, label
%3 = source string of 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECTION. 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).

" Siemens AG 2007 All Rights Reserved

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 address

Definitions: For each address type, it is defined internally how often it may occur in an NC block (for instance, all axes together form one address type for which a block limit also applies).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECTION. 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', ‚ACN', ‚ACP'. 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

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 vocabulary words require that the data in their parameters be written in variables of the type "AXIS". For example, in the vocabulary word POS 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 vocabulary words 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 POS[ZUSTELL]=120 FA[ZUSTELL]=1000

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Key: Press the NC Stop key and select the function "Correction block" with the softkey 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.

" Siemens AG 2007 All Rights Reserved

12050 Channel %1 block %2 DIN address %3 not configured

Parameters: %1 = Channel number
%2 = Block number, label
%3 = NC address in the source text block

Definitions: The name of the NC address (e.g. X, U, X1) is not defined in the control.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Study the Programming Guide and the machine data with respect to the addresses actually configured and their significance and correct the NC 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).
G FUNCTIONS:

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 2007 All Rights Reserved

Definitions: At the text position shown, the grammar in the block is incorrect. Because there are too many possible error possibilities, the precise error case cannot be given.
Example 1:
N10 IF GOTOF ...; The jump condition is missing!
Example 2:
N10 R-50 =12; Incorrect arithmetic parameter number

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 = Invalid parameter in the text

Definitions: The programmed function has been predefined; no parameters are allowed in its call. The first unexpected parameter is displayed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

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 permitted with the valid syntax-determining G function.
Example:
G1 I10 X20 Y30 F1000;
No interpolation parameter may be programmed in the linear block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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, G26Working area, spindle speed limitation
G110, G111, G112 Pole programming with polar coordinates
Example:
G4 F1000 M100: no M function allowed in the G4 block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program G function by itself in the block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12140 Channel %1 block %2 functionality %3 not implemented

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Software construction in the source text

Definitions: In the full configuration of the control functions are possible that are not yet implemented in the current version.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Change the definition of the used variables so that the required operations can be performed.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 change to:R1 = 9 876 543 210.

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 the 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 R1=R2-(-R3); Correct notation
N10 R1=R2--R3; Error!

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the definition of the array, for multi-dimensional arrays, possibly define a second 2-dimensional array and operate with the same array index.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Perform the following tests:
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.

12205 Channel %1 block %2 area specification missing for GUD area

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

Definitions: The area specification
(NCK or CHAN) was not programmed in the definition instruction for a GUD variable.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Complete the area specification for the GUD variable definition in the GUD definition file.
The definition of a GUD variable must conform to the following syntax:
DEF <Area> <Data type> <Variable name> e.g.
DEF NCK INT intVar1
DEF CHAN REAL realVar1

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Select shorter string or divide up the character string into 2 strings. Define a 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"

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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' "

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 1 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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

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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

" Siemens AG 2007 All Rights Reserved

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})

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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. DEF FRAME LOCFRAME = CTRANS(X,200).
Equally, no default values can be programmed for axes in the program run when initializing fields by SET.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Perform 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

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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, vocabulary word, variable, configured identifier.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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

" Siemens AG 2007 All Rights Reserved

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 RS-232 input of NC blocks, because communication between operator panel and NCK is limited to a block length of 242 characters), an alarm is displayed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 unique and may not even appear in the control (exception: local variable).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

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 Y missing!
```

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 with passed parameters do not require any 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 the actual parameters):
N10 XYZ (, Y1, R_TABLE)
Variable X initialized with the value 0
Variable Y is assigned the value of variable Y1 and returns the result after the subroutine pass back to the calling program
Variable A is supplied with the axis in R_TISCH
Variable B omitted!

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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

" Siemens AG 2007 All Rights Reserved

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: The actual parameter and the formal parameter could in principle be different if a conversion would always be possible. In the present case, however, the types are generally not compatible, e.g. STRING (REAL.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Check the transfer parameters of the subroutine call and according to the use, define as call-by-value or call-by-reference parameters.

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 NC.
User-defined functions and procedures:
The number of parameters is established by type and name in the definition.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

" Siemens AG 2007 All Rights Reserved

12360 Channel %1 block %2 dimension of parameter %3 incorrect

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

Definitions: Check the following possible errors:
1.The current parameter is an array, but the formal parameter is a variable
2.The current parameter is a variable, but the formal parameter is an array
-

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remove the value range specification (begins with the vocabulary word 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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.
Summary of the type conversions:

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Define variable type such that the initialization value can be assigned, or select the initialization value appropriate for 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

Definitions: The following causes are possible:
- Invalid index list; an axis index is missing
- Array index does not match the definition of the variable
- An attempt has been made, different to the standard access, to access a variable for the array initialization using SET or REP.
  Access to a single character or omitted indices are not possible.

A nonexistent element was addressed on initializing this array.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 array size. The 1st 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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: -

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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, ...)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Check whether all parameters need to be specified. 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 complete program will be compiled block for block. During this procedure all multiple labels are recognized; this is not always the case with on-line 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

" Siemens AG 2007 All Rights Reserved

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, 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)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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).
GUD data blocks can only cause errors as part of the 'initial.ini download' process.
Cycle program definitions will be reloaded for each Power On/NC-RESET. Thus, the modules can cause errors only in conjunction with this action.
See also the explanations for alarm 6010.

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: In the displayed block, a non-defined G function has been programmed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.
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.

Definitions: %2 = Block number, label.
%3 = G code number.
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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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, ...)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

12481 Channel %1 block %2 program attribute %3 not allowed

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

Definitions: The attribute used in the PROC instruction is not permitted in the current operating mode.
The attribute SAVE, for example, is not allowed in a technology cycle.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press button NC STOP and select the function "Compensation block" using softkey PROGAM CORRECT. The cursor jumps to the incorrect block. Then delete the invalid program attribute.

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 required access authorization has not been set. The required protection level lies outside the permitted value range.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect 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

" Siemens AG 2007 All Rights Reserved

Definitions: The displayed vocabulary word may not be used in this type of block and at this location (all files in the NC 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 the data initialization

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Remove the displayed language elements (vocabulary word) 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 ...

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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 ...

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

" Siemens AG 2007 All Rights Reserved

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
DEFINE G4444 AS G01 G91 G1234
DEFINE M333 AS M03 M50 M99
M17

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Key: Press the NC Stop key and select the function "Correction block" with the softkey 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.

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 200 characters.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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/function not activated

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

Definitions: The identifier displayed has not been defined before being used.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Correct the used name (typing error)
- Check the definition of variables and subroutines
- Check the 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 variable is not known in the control.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 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 and magazines, the corresponding machine data value is set to < 10, or the option 'TM OEM parameters' has not been set.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12553 Channel %1 block %2 name %3 option/function is not active

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

Definitions: The NC function that belongs to this language command is not active. Although the name of the language command is known, the use of this language command in any program is rejected with this alarm.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
• Correct the name used (typing error)
• Activate the NC function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12554 Channel %1 block %2 replacement cycle %3 for the predefined procedure is missing.

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

Definitions: The replacement cycle that is to be called instead of the predefined procedure
is not present / unknown in the control.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC stop key and select the "Compensation block" function by pressing the PROGRAM CORRECT softkey. The cursor will position itself in the faulty block.
- Correct the name used for the predefined procedure (write error)
- Or load the replacement cycle into one of the cycle directories (+ restart)
- Or set the machine data bit for the predefined procedure in $MN_COUPLE_CYCLE_MASK to 0 so that the predefined procedure is executed again.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12555 Channel %1 block %2 function not available (identification %3)

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

Definitions: The identifier is not defined for this system.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
- Correct the name used (typing error)
- For low-level functions, use a more advanced software system
- Check definitions of variables, subroutines and macros
- Declare subroutine with EXTERN, load subroutine to SPF-Dir
- Check interface definition of subroutine

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

12556 Channel %1 block %2 name %3 Name is already known

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

Definitions: The name of the symbol to be created belongs to the scope of the NC language and thus already known. Although the NC function is not active, this name can no longer be used for GUDs, macros and PROC definitions.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct the name used (typing error).

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12573 Channel %1 block %2 motion-synchronous action: Call by reference parameters not allowed %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Source text area

Definitions: Call by reference parameters (keyword VAR) are not possible with technology cycles.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct PROC instruction of technology cycle.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 synchronous 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])

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 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 ...

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.
Example:
PUTFTOCF(1, $AA_OVR, 2, 1, 2)
The variable $AA_OVR is not allowed here.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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]

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program: Use variables of the same type.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program: Replace the on-line variable by an arithmetic variable.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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 are defined in the machine data.
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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.

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.

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

Remedy: Correct INI file or correct MD and create new INI file (via "upload").

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 %3

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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

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

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

" Siemens AG 2007 All Rights Reserved

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.

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

Remedy: Correct part program. If necessary, move parts to a subroutine.

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

12660 Channel %1 block %2 motion synchronous action: variable %3 reserved formotion 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12700 Channel %1 block %2 contour definition programming not allowed as modal sub-programis active

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program

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"

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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 call with G81 - G89 together with an M macro in the block or a G65/G66 macro call together with M macros in the block.
Only one macro or cycle call can appear in an NC block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Divide the cycle and macro calls into several blocks.

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 42162 EXTERN_DOUBLE_TURRET_DIST is 0.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Enter tool clearance for the double turret head in the setting data 42162 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 24100 TRAFO_TYPE_1, 24110 TRAFO_AXES_IN_1[1], 24210 TRAFO_AXES_IN_2[1] are incorrectly set for G07.1, G12.1.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Enter valid transformation identifier for TRACYL in MD 24100 TRAFO_TYPE_1 and the rotary axis number in MD 24110 TRAFO_AXES_IN_1[1] or MD 24210 TRAFO_AXES_IN_2[1].

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

12740 Channel %1 block %2 modal macro call %3 not possible

Parameters: %1 = channel number
%2 = block number, label
%3 = source string

" Siemens AG 2007 All Rights Reserved

Definitions: When calling a modal macro no other modal macro, modal cycle or modal subroutine may be active.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Change part program
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14000 Channel %1 block %2 illegal end of file

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An M02 or an M30 for main programs, an M17 for subroutines, is expected as end of file.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Check whether the program end has been omitted or whether in the last program block a jump has been made to a program section that contains the end identification.
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.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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

14005 Channel %1 block %2 program %3 program-specific start disable has been set

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name
Definitions: Program %3 cannot be executed, as the program-specific start disable has been set for this file.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Reset the program-specific start disable for file %3.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14008 Channel %1 block %2 WRITE command writes in the temporary memory area in /_N_EXT_DIR

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A workpiece is executed from an external data register (Execute from external drives function). The part programs are temporarily stored in the NCK directory /_N_EXT_DIR. An attempt is now made to write in this temporary directory with a WRITE command.
The alarm is intended to indicate that this data is not stored in the original directory on the external data carrier, and will be lost at the next part program selection because the programs in the directory /_N_EXT_DIR will then be deleted.
Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: State a directory that remains permanently loaded in the NCK as the target for the WRITE command (e.g. MPF_DIR).
The alarm can be suppressed with machine data 11415 / $MN_SUPPRESS_ALARM_MASK_2 bit 8.

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

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.

Reaction: 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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: The called program (main program or subroutine) has been called from the currently running part program (main program or subroutine). Either it is not present in the NC memory or the option for the associated function is not set.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct the NC part program.
1.Check the subroutine name in the calling program.
2.Check the name of the calling program.
3.Check whether the program has been transferred to the NC memory.
4.Check or set/correct the options.

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.

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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

Definitions: The selected part program is not contained in the NC memory.

Reaction: Alarm display.

Remedy: Reload the program in the NC memory or check and correct the name of the directory (workpiece overview) and the program (program overview).

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

Definitions: The user does not have any execution rights for the file.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the user rights

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 subroutine call per M or T function: In the block referenced with parameter %2
– an M or T function replacement has already been activated
– a modal subroutine call is already active
– a subroutine return has been programmed
– an M98 subroutine call is active (only in external language mode)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 a subprogram call or return.
The part program must be corrected accordingly.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.
Note:
If a parameter transfer has been programmed via MD 10718 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 constant.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the programming of the M function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14018 Channel %1 block %2 parts program command %3 not executable (protection level setpoint value / actual value: %4)

Parameters: %1 = Channel number
%2 = Block number, Label
%3 = Programmed command
%4 = Protection level of the command
%5 = Current protection level

Definitions: A protection level assigned to the part program command %3 is logically higher (smaller value) than the current access right or the command does not exist in the current control configuration.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program. The valid language commands for the associated system configuration are described in the Siemens Programming Guide or the manufacturer's documentation.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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.

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

Remedy: Change part program.

" Siemens AG 2007 All Rights Reserved

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

14022 Channel %1 block %2 error on function or procedure call, error code %3

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

Definitions: An error occurred during a function or procedure call.
The cause of the error is indicated more closely by an error code.
The meaning of the error code can be found in the documentation of the function or procedure that caused the error.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the 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.

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

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

Definitions: An FCTDEF command was programmed with a polynomial number that exceeds the maximum value set in $MC_MM_NUM_FCTDEF_ELEMENTS.

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

Remedy: Modify part program.

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

14027 Channel %1 block %2 motion-synchronous action: Too many technology cycles programmed.

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

Definitions: You can call a maximum of eight technology cycles with one motion-synchronous action. You exceeded the upper limit.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

14028 Channel %1 block %2 motion-synchronous action: Technology cycle programmed with too many parameters

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

Definitions: Maximum number of transfer parameters for one technology cycle exceeded.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change technology cycle

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14030 Channel %1 block %2 combine OSCILL and POSP during oscillation with infeedmotion

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 2007 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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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 NC 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 MD 21000 CIRCLE_ERROR_CONST divided by MD 21010 CIRCLE_ERROR_FACTOR), or
• greater than the programmed radius multiplied by the MD 21010 CIRCLE_ERROR_FACTOR (for large radii, if the programmed radius is greater than the quotient of the MD 21000 CIRCLE_ERROR_CONST divided by MD 21010 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.
Check MD 21000 CIRCLE_ERROR_CONST and MD 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change 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.

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

Remedy: Change part program

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

14055 Channel %1 block %2 impermissible NC language substitution, error code %3

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

Definitions: This alarm occurs in conjunction with an NC language substitution configured in $MA_AXIS_LANG_SUB_MASK. Error code %3 gives more detailed information about the cause of the problem:
Error code:
1: Several events had been programmed, causing the replacement cycle to be called. Only one substitution is allowed per part program line.
2: A non-modal synchronized action had also been programmed for the part program line with the NC language substitution.
3: The system variables $P_SUB_SPOSIT and $P_SUB_SPOSMODE were called outside a replacement cycle.

Reaction: Correction block is reorganized.
Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Modify the NC 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: A skip level larger than 1 is specified for "differential block skips" (in package 1, the specification of a value for the skip level will already be rejected by the compiler as syntax error, namely, only one "suppress block" level ON/OFF is possible).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Enter skip level (the number behind the slash) 1.

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

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

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Check for the following possible errors in the NC part program:
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 the program part loop with CALL <programname> BLOCK <startlabel> TO <endlabel> was not found or the same program part loop was called recursively.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Check the start and end label for the program loop 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

Definitions: The instruction 'TML()' may only be used in the subprogram, which replaces the T command.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

" Siemens AG 2007 All Rights Reserved

Remedy: Change 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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Change part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14090 Channel %1 block %2 invalid 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
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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

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
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Index = 1: Substitute "RET" command with M17/M30.
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:
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the program section to avoid exceeding the value range, for example by using a different 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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
Choose the parameter string appropriate for the axis name, possibly change the axis name in the machine data. (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.

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

Remedy: Change part program. If this is an input, the ISNUMBER built-in function (with the same parameter) can be used to test whether the string represents a number.

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

" Siemens AG 2007 All Rights Reserved

14099 Channel %1 block %2 result in string concatenation too long

Parameters:

Definitions: The result of string chaining returns a string which is greater than the maximum string length laid down by the system.

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

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: Up to 4 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Press the NC Stop key and select the function "Correction block" with the softkey 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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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.

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

Remedy: Reduce the number of initialization values.

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.

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

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.

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

Remedy: Modify the program.

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

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.

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

Remedy: Modify the program.

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

Reaction: Correction block is reorganized.
Interpreter stop
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Modify program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.
Reaction: Correction block is reorganized.
Interpreter stop
Interface signals are set.
Alarm display.
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 for the tool length compensation with H word and G43/G44 in the ISO M mode using the MD 20380 TOOL_CORR_MODE_G43G44 the variant C is activated (the tool length acts in the programmed axis), just one geometry axis together with H, if the MD 20384 TOOL_CORR_MULTIPLE_AXES is not set. The alarm is then issued when either no geometry axis or more than one geometry axis has been programmed together with H .
The programming of multiple axis is permitted when the MD 20384 TOOL_MULTIPLE_AXES = TRUE has been set.
If no axis is specified, this is always an error.
Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.
Remedy: Change MD 20380 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 selected H number %3 does not match tool %4

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: If a tool compensation is activated in the ISO M language mode (G43/G44), a tool number (H) must be specified.
Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.
Remedy: Change 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

" Siemens AG 2007 All Rights Reserved

Definitions: If tool compensation (G43/G44) is activated in the ISO M language mode, the linear type of interpolation must be active.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14180 Channel %1 block %2 H number %3 is not defined

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

Definitions: The specified H number is not assigned to any tool (ISO M).

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14185 Channel %1 block %2 D number %3 is not defined

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

Definitions: The specified D number is not assigned to any tool (ISO M).

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14196 Channel %1 block %2 error %3 on interpreting the contents of $SC_CUTDIRMOD

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

Definitions: An error has occured during the interpretation of the strings contained in setting data $SC_CUTDIRMOD. This setting data is always read when a new edge is selected. The error code indicates the cause of the error:
1: The string only consists of blanks or a sign
2: Unknown frame name after $P_
3: No colon after the first valid frame name
4: Insufficient memory space for creating a frame internally
5: Invalid frame index
6: Further characters found after complete string
7: Second frame name is missing after the colon
8: Impermissible frame rotation (surface normals are rotated against each other by 90 degrees or more)
9: Invalid frame chain (the first frame must come before the second frame in the frame chain)
10: Invalid axis name
11: Axis is not a rotary axis
12: Invalid string that cannot be assigned to any of the error types 1 to 11
20: Invalid angle statement (numerical value)
30: Invalid angle of rotation (not an integer multiple of 90 degrees)

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

Remedy: Enter valid string in setting data $SC_CUTDIRMOD.

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

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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).

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change the 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 20360 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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Change the part program
Reset bit 2 in the MD 20360 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

Definitions: In the endpoint specification of a traversing block with G00, G01, G02 or G03 in polar coordinates, the polar radius entered for the vocabulary word 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 of the plane
G111 ... Zero point of the current WCS
G112 ... last pole

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct the NC part program – valid inputs for the polar radius are only positive absolute values that specify the distance between the current pole and the block end point (the direction is defined with the polar angle AP=...).

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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 vocabulary word 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 ... On the last programmed point of the plane
G111 ... On the zero point of the current workpiece coordinate system (WCS)
G112 ... On the last pole

Reaction: 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 to +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 vocabulary word 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 of the plane
G111 ... Zero point of the current WCS
G112 ... last pole

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct the NC part program – valid inputs for the polar radius are only positive absolute values that specify the distance between the reference point and the new pole (the direction is defined with the polar 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 vocabulary word 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).

" Siemens AG 2007 All Rights Reserved

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 of the plane
G111 ... Zero point of the current WCS
G112 ... last pole

Reaction: 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.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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,...).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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 axis,
• 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. G group incorrect (only G01 to CIP).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

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

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.

14320 Handwheel %1 used twice (%2) in channel %3 axis %4

Parameters: %1 = Handwheel number
%2 = Use
%3 = Channel
%4 = Axis

Definitions: Warning that the designated handwheel has been used more than once:
The second parameter supplies the explanation:
1: Block with axial handwheel override for this axis cannot be performed for this axis because the handwheel for the axis is performing a DRF motion
2: Block with speed override for the path cannot be performed because the handwheel for the axis of the path is performing a DRF motion
3: Block with contour handwheel cannot be performed because the handwheel for the axis of the path is performing a DRF motion
4: PLC axis with axial handwheel override cannot be started immediately because the handwheel for the axis is performing a DRF motion
5: The axis is a reciprocating axis with axial handwheel override, the oscillating motion cannot be started immediately because the handwheel for the axis is performing a DRF motion
6: The DRF motion for this axis cannot be performed because an handwheel override for this axis with the handwheel is active
7: The DRF motion for this axis cannot be performed because a speed override of the path with the handwheel is active and the axis belongs to the path
8: The DRF motion for this axis cannot be performed because the contour handwheel with this handwheel is active and the axis belongs to the path
9: The DRF motion for this axis cannot be performed because the axis is a PLC axis with handwheel override that is active with the handwheel
10: The DRF motion for this axis cannot be performed because the axis as reciprocating axis with handwheel override is active with this handwheel

Reaction: Alarm display.

Remedy: Use the handwheel for only one purpose.

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

14401 Channel %1 block %2 transformation not available

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

Definitions: The required transformation is not available.
Example:
The following has been programmed: N220 TRACYLI(3) ;Transformation no. 3-ON
However, only transformation 1 and 2 applies.

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

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

14403 Channel %1 block %2 preprocessing and main run might not be synchronized

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

Definitions: Positioning axis curves cannot be predicted reliably. 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.

Reaction: Alarm display.

Remedy: Change 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.
Possible error causes can be:
An axis traversed by the transformation has not been enabled:
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)
Transformation-dependent error causes can be:
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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.
Change the part program or change the machine data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the part program, use TANGDEL to delete the active tangential control.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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_FOR_CORRPOS_NOTALLOWED.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Modify part program. Change machine data CORR_OR_AXIS_NOT_ALLOWED.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14430 Channel %1 block %2 tangential axis %3 must not be traversed as POS axis

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

Definitions: A tangential corrected axis cannot be traversed as positioning axis.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the part program, use TANGDEL to delete the active tangential control.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14432 Channel %1 block %2 rounding length for tangential axis %3 is zero.

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

Definitions: A smoothing length for the activation of the tangential control with TANGON() for a tangential axis coupled in the preparation, otherwise any discontinuities of the tangential axis cannot be smoothed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14434 Channel %1 block %2 rel. lift-off path for tangential axis %3 is invalid

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

Definitions: The factor r for the relative lift-off path programmed for TLIFT must lie in the range 0 =< r < 1.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Define the subroutine in accordance with the type used.
1.Usual subroutine structure (without parameter transfer):
% SPF 123456
:
M17
2.Subroutine structure with vocabulary word and subroutine name (without parameter transfer):
PROC UPNAME
:
M17
ENDPROC
3.Subroutine structure with vocabulary word and subroutine name (with parameter transfer "call-by-value"):
PROC UPNAME (VARNAME1, VARNAME2, ...)
:
M17
ENDPROC
4.Subroutine structure with vocabulary word 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify NC part program appropriately.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

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

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

Definitions: The reload buffer cannot be set up while loading the INITIAL_INI block, because the RAM of the NC does not have sufficient memory space.

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

Remedy: Make available free memory space in the NC area, for example, by deleting part programs that are not longer required.

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 "processing from external source" cannot be deleted, possible cause:
- HMI PLC communication was not terminated.

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

Remedy: Power On clears all reload buffers.

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

14602 Channel %1 block %2 timeout while reloading from external.

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 MD 10132 MMC_CMD_TIMEOUT.

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

Remedy: Increase MD 10132 MMC_CMD_TIMEOUT.

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

" Siemens AG 2007 All Rights Reserved

14603 Channel %1 block %2 timeout during execution from external source.

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

Definitions: If a program is selected for execution from external source, it will be expected that the first part program line can be read from the reload buffer within 60s after part program start.
Otherwise, part program processing will be aborted with alarm 14603 due to the assumption that the connection to the HMI or the external device is faulted.

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

Remedy: Check the connection to the HMI and repeat selection of the program that is to be executed from external source.

Program Continuation: Clear alarm with the RESET key. Restart part program
- Acknowledge the alarm by pressing the RESET key
- Repeat program selection
- Start the 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.

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

Remedy: Abort program with reset.
• Correct program on MMC or PC.
• Restart reloading (possibly with block search for interrupt location).

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

14615 Channel %1 An error occurred while handling the function syntax check: identifier %3

Parameters: %1 = Channel number
%2 = Block number
%3 = Error code

Definitions: An error occurred while handling the function syntax check via the PI services _N_CHKSEL, _N_CHKRUN, _N_CHKABO and _N_SEL_BL. Parameter %3 describes the error situation more closely:
Value
1: An invalid line number was transferred with the PI service _N_SEL_BL
2: An invalid line number for the range end was transferred with the PI service _N_CHKRUN
3: PI service _N_CHKSEL was activated although a block selection (PI service _N_SEL_BL) was active for the selected program.

Reaction: Alarm display.

Remedy: Value
1: Supply PI service _N_SEL_BL with the correct line number
2: Supply PI service _N_CHKRUN with the correct line number for the range end
3: Ensure that the channel is in reset status before activating the PI service _N_CHKSEL.

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

" Siemens AG 2007 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 for control-internal commands such as part program selection, reset or change configuration-specific machine data.

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

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) error-free execution is possible by repeating the program or operator action.
Otherwise, the A&D MC system support should be contacted with a precise description of the error situation:
SIEMENS AG, After-Sales Support for A&D MC Products, Hotline
(Phone: see page 1-9)

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 blocks not available

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.

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

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 = Index for section

Definitions: The start-up and reset of the control produces initialization blocks. This can cause errors because of incorrect machine data settings.
The parameter %3 specifies the section of the initblock generation in which the error occurred:
Section 0: Error during synchronization preprocessing/main run
Section 1: Error on selection of tool length compensation.
Section 2: Error on selection of transformation.
Section 3: Error on selection of zero offset.
The cycle interfaces are also read in during the power-up procedure. If an error occurs here, it will be reported with "section 5".

Reaction: Interpreter stop
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: For section 0-3: Load the standard machine data
For section 5: Reload the cycles

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

" Siemens AG 2007 All Rights Reserved

14711 Channel %1 transformation selection not possible as axis %2 not available

Parameters:

Definitions: The configuration of the machine data 20110 RESET_MODE_MASK and 20140 TRAFO_RESET_VALUE means that a reset or a control power-up should select a transformation. This, however, is not possible, because the required axis %2 is not available. Possible causes: The axis is used by the PLC.

Reaction: Interface signals are set.
Alarm display.

Remedy: Select transformation using a part program command.

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

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.

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

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.

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

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.

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

" Siemens AG 2007 All Rights Reserved

Remedy: - Modify part program.
- Check tool data.
- Check machine data.

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

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.

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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: Processing of motion synchronous actions requires resources that are configured via the machine data $MC_MM_IPO_BUFFER_SIZE, $MC_MM_NUM_BLOCKS_IN_PREP, $MC_MM_NUM_SAFE_SYNC_ELEMENTS and $MC_MM_NUM_SYNC_ELEMENTS. If these resources are insufficient for executing the part program, then this alarm is issued. 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,
$MC_MM_NUM_SAFE_SYNC_ELEMENTS.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct part program or increase resources.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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".

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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".

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.
- RET allowed in technology cycle only
- Function "Several feeds" not allowed in technology cycle
- H and M function outputs not allowed with WHENEVER, FROM and DO
- MEASA / MEAWA / MEAC with WHENEVER, FROM and DO not allowed
- DELDTG and STOPREOF allowed only in blockwise synchronous action with WHEN and EVERY

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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-variable auxiliary functions has exceeded the maximum permitted number. The number is set in MD 28150 MM_NUM_VDIVAR_ELEMENTS.

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

Remedy: Modify part program or machine data.

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

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 NC system settings has been defined for all auxiliary functions by means of the MD 11100 AUXFU_MAXNUM_GROUP_ASSIGN (default: 1).
For each user-defined auxiliary function to be assigned to a group, the assignment is made using 4 machine data items.
MD 22010 AUXFU_ASSIGN_TYPE: Auxiliary function type, e.g. M
MD 22000 AUXFU_ASSIGN_GROUP: Required group
MD 22020 AUXFU_ASSIGN_EXTENSION:Possibly required extension
MD 22030 AUXFU_ASSIGN_VALUE: Function value

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

14780 Channel %1 block %2 unreleased option used (identification %3)

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

Definitions: A non-released option has been used in the block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program, retrofit option.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14782 Channel %1 block %2 non-active function used (identification %3)

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

Definitions: An inactive function is used in the block.
Flag, short description
1 Transformation
2 Tool H numbers

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.
Activate function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14783 Channel %1 block %2 coordinate system-specific working area limitation is not active

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

Definitions: An attempt is made in the block to activate a group of the coordinate system-specific working area limitation.
However, this group is not set up (see machine data $MC_MM_NUM_WORKAREA_CS_GROUPS).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
The NC program is stopped. The G code of the group WALCS01 - WALCS10 can be changed.

Remedy: - Modify part programm.
- Activate more coordinate system-specific working area limitations.

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

Definitions: In the NC block, an axis has been programmed that is already being traversed by the PLC.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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 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.

Reaction: 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Adjust the value range in accordance with the Programming Guide.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change the part program or change the machine data.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 vocabulary word LIMS=.... The values are in the range 0.1 - 999 999.9 [rev/min].

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program the maximum spindle speed for the constant cutting speed within the limits given above. The vocabulary word 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 GWPS programming (constant grinding wheel surface speed) 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.
- An attempt has been made to select a tool which has not been defined.
- An attempt has been made to select an edge (implicitely) which has not been defined (implicit selection: D1 of a tool, if no tool has been specified.)
- 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.
- Selection refers to an invalid spindle number.
- A grinding wheel radius equal to zero has been specified.

On deselecting GWPS 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.
- Deselection refers to an invalid spindle number.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.
- 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 a tool that has not been defined.
- An attempt has been made to select an edge (implicitely) that has not been defined. (Implicit selection: D1 of a tool, if no edge has been specified.)
- 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 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)
A grinding wheel radius equal to zero has been 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.
- Deselection refers to an invalid spindle number.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14830 Channel %1 block %2 wrong feed type selected

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

Definitions: G97 has been programmed in the displayed block although G96 was not (or G97 already) active previously.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Remove G97 from the displayed block and program the correct feed type (G93, G94, G95 or G96) for the machining section which follows.

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].

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

14850 Channel %1 block %2 changing the reference axis for a constant cutting speed not allowed

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

Definitions: The attempt was made via the SCC[AX] instruction to change the reference axis for a constant cutting speed.
This is not allowed if the indicated axis is no geometry axis.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Please inform authorized personnel/service.
When programming SCC[AX] indicate a geometry axis known in the channel.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Select the programming variant for which the dimension can be reliably taken from the workpiece designation (avoidance of conversion 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 all 3 points (start, 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 start and end points.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 start 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.

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.

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

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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:
Reduce log file size requirement by:
Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops STOPRE.

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.

Reaction: 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:
Reduce log file size requirement by:
Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops STOPRE.

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

15120 If a power failure occurs now, data changed before will be lost; buffer size = %1

Parameters: %1 = Buffer size

Definitions: Notification alarm. The alarm has no negative impact on the current machining.
The system-internal data buffer in which the last buffered data are stored that were changed, has overflown (as the data change rate is currently too high).
The alarm warns of a spontaneous power failure in this situation
(mains fault, disconnect the system from the power supply) that would cause a loss of buffered data changed beforehand (tool data, parts programs,
R parameters, GUDs,...
If the system is operated in an environment in which a power failure cannot occur, output of this alarm can be avoided via machine date $MN_MM_ACTFILESYS_LOG_FILE_MEM = 0.
Parameter %1 informs about the buffer size set.

Reaction: Alarm display.

Remedy: If the alarm is only present sporadically, it can be regarded as a notification only.
The regular control behavior is not influenced.
If the alarm is permanently present, please inform the authorized personnel/service.

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

" Siemens AG 2007 All Rights Reserved

15122 Power ON after power failure: %1 data were restored, of which %2 machine data, %3 errors.

Parameters: %1 = Number of data
%2 = Number of machine data
%3 = Number of errors occurred

Definitions: Notification alarm. The alarm has no negative effect as long as %3 the number of errors occurred is zero.
%1 indicates the number of elementary and complex data restoring steps which were taken after a power OFF during power ON orduring a power failure to restore the persistent NCK data.
%2 indicates the number of restored machine data. If the value is larger than zero, another warm restart (NCK reset) may be necessary to make the - possibly configuring - machine data changes prior to the power failure effective.
%3 indicates the number of errors occurred during data restoring.

Reaction: Alarm display.

Remedy: As long as %3 number of errors occurred is zero, the alarm is only informative.
As long as %3 number of errors is larger than zero, the alarm indicates a software error.
Continuing with the data is not recommended.
Please install a suitable archive file before continuing to avoid follow-up problems.
Please inform authorized personnel/service.

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

15150 Channel %1 block %2 reload from external aborted

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

Definitions: The processing of extern has been terminated because the reload buffer does not contain sufficient machine function blocks (traversing blocks, auxiliary function, delay 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.

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

Remedy: Add machine function blocks to the part program.
Increase the size of the reload buffer (MD 18360 MM_EXT_PROG_BUFFER_SIZE).

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.

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

Remedy: Please inform the authorized personnel/service department.

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

" Siemens AG 2007 All Rights Reserved

15165 Channel %1 block %2 error when translating or interpreting Asup %3

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

Definitions: At part program start and at start of an ASUB under Reset condition, the relevant data of all the ASUBs that can be activated at that time are preprocessed:
- PLC ASUBs
- With $MC_PROG_EVENT_MASK configured event-controlled program calls
- ASUB after block search ($MN_SEARCH_RUN_MODE bit 1=1)
- Editable system ASUB ($MN_ASUP_EDITABLE)
If an error occurs (converter or interpreter), alarm 15165 will be output first and then a converter or interpreter alarm that describes more details of the error. Alarm 15165 will cause an interpreter stop. A compensation block will not be possible.

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

Remedy: Modify part program.

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.

Reaction: Alarm display.

Remedy: Correct part program.

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

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.

Reaction: Alarm display.

Remedy: Correct part program.

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

15180 Channel %1 block %2: Error on editing program %3 as INI/DEF file.

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

Definitions: An error occurred while reading the INI file. The error message refers to the program specified here.

Reaction: Alarm display.

Remedy: Correct part program.

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 detected while processing an INI file.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Please inform the authorized personnel/service department. Correct INI file or correct MD and create new INI file (via "upload").

Program Continuation: Switch control OFF - ON.

15186 Channel %1 %2 errors in GUD, macro or INI file

Parameters: %1 = Channel number
%2 = Number of detected errors

Definitions: %2 errors were found when processing GUD/macro definition files (DEF files) or initialization files (INI files)
Alarm 15180 has already informed about the corresponding file.
Prior to that the errors shown were reported by error-specific alarms, e.g. 12080 "syntax error".

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

Remedy: Modify definition file or initialization file

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

15187 Channel %1 error during execution of PROGEVENT file %2.

Parameters: %1 = Channel number
%2 = PROGEVENT file name

Definitions: An error has occurred on executing PROGEVENT.
With alarm 15187, the name of the program that was started as PROGEVENT is displayed. Alarm 15187 is displayed together with the alarm that describes the error cause. Alarm 15187 is also output when the alarm occurs in a subroutine started from PROGEVENT.

Reaction: Interface signals are set.
Alarm display.

Remedy: Correct the PROGEVENT file (subroutine).

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

15188 Channel %1 error during execution of ASUB file %2.

Parameters: %1 = Channel number
%2 = ASUB file name

Definitions: An error has occurred on executing an ASUB.
Alarm 15188 displays the name of the program that was started as ASUB. Alarm 15188 is output together with the alarm that describes the error cause. Alarm 15188 is also output when the alarm occurs in a subroutine started from the ASUB.

Reaction: Interface signals are set.
Alarm display.

Remedy: Correct the ASUB file (subroutine).

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program the STOPRE preprocessing stop before the subroutine call.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

Reaction: Alarm display.

Remedy: Correct the specified search target accordingly or reload the file.

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

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. Valid block search commands are:
TypeMeaning
1Search for block number
2Search for label
3Search for string
4Search for program name
5Search for line number in a file

Reaction: Alarm display.

Remedy: Change 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".

Reaction: 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.

Reaction: Alarm display.

Remedy: Repeat the input with corrected label.

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

" Siemens AG 2007 All Rights Reserved

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.

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

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

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.

Reaction: 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).

Reaction: 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: A transformation change occurred after "search at block end".
The position accumulated during the search cannot be traversed incrementally.

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

Remedy: Find search destination in which the axes are programmed using an absolute reference.
Deactivate adding of the accumulated search position with SD 42444 TARGET_BLOCK_INCR_ = FALSE.
Use search run with calculation "at contour".

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 NC range or
2.Does not have the protection level required to perform the function

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
Check whether the selected INI block is contained in the file system of the NC. 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.

" Siemens AG 2007 All Rights Reserved

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.

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

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).

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 interpreter has detected an invalid instruction (e.g. a traverse instruction) while processing of an Init module.

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

Remedy: Remove all motion actions and auxiliary 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).

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

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

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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

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

" Siemens AG 2007 All Rights Reserved

15700 Channel %1 block %2 illegal cycle alarm number %3

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

Definitions: A SETAL command with a cycle alarm number smaller than 60,000 or larger than 67,999 has been programmed.
Alarm Reactions of Siemens standard cycles:
No. 61 000 - 61 999: Interpreter stop; delete with reset
No. 62 000 - 62 999: Compensation block; delete with NC Start

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

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: The starting conditions are faulty for the contour preparation (CYCLE 95):
G40 (deselection of the tool radius compensation) is not active

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

Remedy: Change part program: Deselect tool radius compensation with G40.

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 column number for a contour table has a fixed size. The current value is contained in the current Technology Programmer's Guide.

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

Remedy: Correct the array definition for the contour table.
The number of rows can be defined freely and corresponds to the number of contour elements (circles, straight lines). The number of columns is fixed (see the Cycle Programming Guide).

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: A non-existent touch probe for measuring with deletion of remaining distance has been selected in the part program.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

15910 Channel %1 block %2 touch probe not allowed

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

Definitions: A non-existent touch probe for measuring without deletion of remaining distance has been selected in the part program.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

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: In the part program, measurement with deletion of remaining distance without axis details has been selected or the programmed traversing motion is zero.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

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: In the part program, measurement without deletion of remaining distance without axis details has been selected or the programmed traversing motion is zero.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change part program.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

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.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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: A block is to be repositioned for which repositioning information exists.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: If necessary, change the part program.
Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

16025 Channel %1 block %2 impermissible axis change in REPOS command by axis %3.

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

Definitions: With the REPOS command, an axis or spindle was programmed that was in the NEUTRAL state at that time. As the REPOS command cannot execute any implicit GET, these axes/spindles cannot be repositioned. Part program editing is therefore aborted.

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

Remedy: Assign the axes/spindles that are to be repositioned to the channel via GET command prior to the REPOS command.
Example:
GET(A); assign the A axis to the channel
REPOSL A; reposition the geometry axes and A axis

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 spindle function.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 MD 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 SD 42800 SPIND_ASSIGN_TAB[].

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

Remedy: Correct the setting data or change the 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

" Siemens AG 2007 All Rights Reserved

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)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

16111 Channel %1 block %2 spindle %3 No speed programmed

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

Definitions: A speed must be programmed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program speed with S[spindlenumber]=...

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 control.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Do not program spline and polynomial interpolation.

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 represented on any machine axis.
Example:
Polar coordinate system with X, Z, and C axis

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Do not program the associated axis as a geometry axis.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Traverse geometry axes as positioning axes only with rotation deactivated. Deactivate rotation: Vocabulary word 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
%3 = Axis name, spindle number

Definitions: A rotation of a geometry axis which does not exist was programmed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Change 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

Definitions: A negative chamfer or rounding has been programmed under the vocabulary words CHF= ..., RND=... .

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

16510 Channel %1 block %2 no facing axis for diameter programming available

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

Definitions: Diameter programming has been activated with the vocabulary word 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 MD 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.

16520 Channel %1 axis %2, diameter programming active, function %3 is not executed

Parameters: %1 = Channel
%2 = Axis, spindle
%3 = NC function

Definitions: The function is not executed with diameter programming active for the stated axis.
The following functions may be affected:
1 - Axis interchange
2 - Axis container rotation

Reaction: Interpreter stop
Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Disable diameter programming of the axis before activating the function.

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

16600 Channel %1 block %2 spindle %3 gear stage change not possible

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

Definitions: The programmed speed is outside the speed range of the set gear stage. In order to execute the programmed speed, the gear stage must be changed. In order to be able to execute the automatic gear stage change (M40 is active), the spindle must be in speed control operation.
>The alarm will no longer be output after having set bit 30 (0x40000000) in MD 11410 SUPPRESS_ALARM_MASK. However, the function will not be affected by this.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: The changeover to speed control operation is performed by programming M3, M4 or M5. The M functions can be written together with the S word in the same block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16670 Channel %1 block %2 following axis/spindle %3 maximum number of CP modules (%4) has been exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Max. number of CP modules

" Siemens AG 2007 All Rights Reserved

Definitions: An attempt was made to activate more generic couplings than are configured in MD18450.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Reduce the number of defined or active couplings, or increase the number of coupling modules configured in MD18450.
If necessary, buy another option stage of the generic coupling.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16671 Channel %1 block %2 following axis/spindle %3 maximum number of CP modules (%4) has been exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Max. number of CP modules

Definitions: An attempt was made to activate more generic couplings than are configured in MD18450.

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

Remedy: Reduce the number of defined or active couplings, or increase the number of coupling modules configured in MD18450.
If necessary, buy another option stage of the generic coupling.

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

16672 Channel %1 block %2 leading axis/spindle %3 maximum number of CP master values (%4) exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Max. number of CP master values

Definitions: An attempt was made to activate more master values of generic couplings than are configured in MD18452.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Reduce the number of defined or active master values, or increase the total number of master values configured in MD18452.
If necessary, buy another option stage of the generic coupling.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16673 Channel %1 block %2 leading axis/spindle %3 maximum number of CP master values (%4) exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Max. number of CP master values

Definitions: An attempt was made to activate more master values of generic couplings than are configured in MD18452.

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

Remedy: Reduce the number of defined or active master values, or increase the total number of master values configured in MD18452.
If necessary, buy another option stage of the generic coupling.

" Siemens AG 2007 All Rights Reserved

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

16678 Channel %1 block %2 following axis/spindle %3 status %4 impermissible traversing instruction

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Status

Definitions: An addtional traversing in the following axis/spindle is not permitted in the current status of the generic coupling.
Example: CPOF=X G0 X100 is not permitted.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.
A motion in the following axis/spindle can be programmed with CPFPOS, CPON or CPOF

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16680 Channel %1 block %2 following axis/spindle %4 instruction %3 programmed repeatedly

Parameters: %1 = Channel number
%2 = Block number, label
%3 = CP instruction
%4 = Axis name, spindle number

Definitions: The stated instruction has been programmed repeatedly in the block for the same following axis/spindle of a generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16682 Channel %1 block %2 following axis/spindle %3 instructions %4 are not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = CP instruction

Definitions: The stated instructions are not permitted together in one block for a following axis/spindle of a generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16684 Channel %1 block %2 following axis/spindle %3 instructions %4 are not possible separately.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = CP instructions

" Siemens AG 2007 All Rights Reserved

Definitions: The stated instructions are only permitted together in one block for a following axis/spindle of a generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16686 Channel %1 block %2 following axis/spindle %3 type of coupling/instruction %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = CP instructions

Definitions: The stated instruction is not permitted for the stated type of generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16687 Channel %1 block %2 following axis/spindle %3 type of coupling/instruction %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = CP instructions

Definitions: The stated instruction is not permitted for the stated type of generic coupling.

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

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16688 Channel %1 block %2 following axis/spindle %3 coupling type %4 maximum number of master values exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Coupling type

Definitions: The maximum number of master values has been exceeded for the stated type of generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program, reduce number of master values or use a different type of coupling.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16689 Channel %1 block %2 following axis/spindle %3 coupling type %4 maximum number of master values exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Coupling type

" Siemens AG 2007 All Rights Reserved

Definitions: The maximum number of master values has been exceeded for the stated type of generic coupling.
Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: Modify part program, reduce number of master values or use a different type of coupling.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16690 Channel %1 block %2 following axis/spindle %3 changing the reference system %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Reference system
Definitions: An attempt was made to change the reference system with generic coupling active.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part programm.
End coupling and reactivate with desired reference system.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16691 Channel %1 block %2 following axis/spindle %3 changing the reference system %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Reference system
Definitions: An attempt was made to change the reference system with generic coupling active.
Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: Modify part programm.
End coupling and reactivate with desired reference system.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16692 Channel %1 block %2 following axis/spindle %3 maximum number of couplings in the block %4 has been exceeded

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Maximum number of couplings
Definitions: The maximum number of generic couplings in the block has been exceeded.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Reduce the number of generic couplings programmed in the block.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

16694 Channel %1 block %2 following axis/spindle %3 status/instruction %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Status, instruction

Definitions: The stated instruction is not permitted for the current status of the generic coupling.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16695 Channel %1 block %2 following axis/spindle %3 status/instruction %4 is not possible.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Status, instruction

Definitions: The stated instruction is not permitted for the current status of the generic coupling.

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

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16696 Channel %1 block %2 following axis/spindle %3 coupling has not been defined.

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

Definitions: An instruction to an undefined coupling is to be executed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part programm.
Define the coupling and activate, if necessary, before the instruction.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16697 Channel %1 block %2 following axis/spindle %3 coupling has not been defined.

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

Definitions: An instruction to an undefined coupling is to be executed.

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

Remedy: Modify part programm.
Define the coupling and activate, if necessary, before the instruction.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

16698 Channel %1 block %2 following axis/spindle %3 leading axis/spindle %4 has not been defined.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: An instruction to an undefined leading axis/spindle of a coupling is to be executed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part programm.
Define the leading axis/spindle and activate, if necessary, before the instruction.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16699 Channel %1 block %2 following axis/spindle %3 leading axis/spindle %4 has not been defined.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: An instruction to an undefined leading axis/spindle of a coupling is to be executed.

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

Remedy: Modify part programm.
Define the leading axis/spindle and activate, if necessary, before the instruction.

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.
G33 (thread with constant lead) and the feed have not been programmed with G94 or G95.
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).
G331 or G332 (rigid tapping) and the feed have not been programmed with G94.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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

Remedy: Add S value or direction of rotation for the master spindle in the displayed block.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program M5 or SPOS 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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

Example:
N100 G33 Z400 K2; Thread lead 2 mm, thread
: end Z=400 mm
N200 SPOS=0; Transfer spindle to axis operation
N201 G90 G331 Z-50 K-2; Tapping at Z=-50, counter-clockwise rotation
N202 G332 Z5; Retraction, automatic direction reversal
N203 S500 M03;Spindle reverts to spindle operation

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Specify geometry axis and corresponding interpolation parameters.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16746 Channel %1 block %2 spindle %3 selected gear stage %4 not installed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Spindle number
%4 = Gear stage

Definitions: The first gear stage data block is active. The required gear stage is not installed in the 1st gear stage data block. The number of gear stages installed is configured in machine data 35090 $MA_NUM_GEAR_STEPS.
Examples of the occurrence of the alarm with 3 three gear stages installed (MD 35090 $MA_NUM_GEAR_STEPS = 3):
* ... M44 or M45 has been programmed for the spindle concerned
*...M70 has been programmed and machine data 35014 $MA_GEAR_STEP_USED_IN_AXISMODE is larger than 3.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program: Only those valid gear stages can be entered which have also been installed according to machine data MA_NUM_GEAR_STEPS.
Limit M70 configuration (MD 35014 $MA_GEAR_STEP_USED_IN_AXISMODE) to MD 35090 MA_NUM_GEAR_STEPS.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16747 Channel %1 block %2 spindle %3 inserted gear stage %4 for tapping not installed

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Spindle number
%4 = Gear stage

Definitions: The second gear stage data block has been activated for tapping with G331.
However, the current gear stage has not been installed in the second gear stage data block. The number of gear stages installed is configured in machine data 35092 NUM_GEAR_STEPS2. The gear stage cannot be changed in traversing blocks. The gear stage appropriate for the speed must be loaded before the traversing block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Procedure for automatically engaging the suitable gear stage prior to thread cutting:
* Program the spindle speed (S) in a G331 block without axis motions and prior to thread cutting, e.g. G331 S1000.
* Activate M40 for the spindle.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

16748 Channel %1 block %2 spindle %3 gear stage %4 expected

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Spindle number
%4 = Gear stage

Definitions: G331 activates the second gear stage data block for tapping.
The programmed speed (S) of the master spindle lies outside the speed range of the active gear stage in the current traversing block.
The gear stage cannot be changed in the traversing block. The gear stage appropriate for the speed must be loaded prior to the traversing block.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Procedure for automatically engaging the suitable gear stage prior to thread cutting:
* Program the spindle speed (S) in a G331 block without axis motions and prior to thread cutting, e.g. G331 S1000.
* Activate M40 for the spindle.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Do not write instruction.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

16757 Channel %1 block %2 for following spindle %3 coupling as leading spindle/axis already existing

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

Definitions: A coupling has been switched on in which the following spindle/axis has already been active as leading spindle/axis in another coupling. Chained couplings cannot be processed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Check in the parts program whether the following spindle/axis is already active as leading spindle/axis in another coupling.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16758 Channel %1 block %2 for leading spindle %3 coupling as following spindle/axis already existing

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

Definitions: A coupling has been switched on in which the leading spindle/axis has already been active as following spindle/axis in another coupling. Chained couplings cannot be processed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Check in the parts program whether the leading spindle/axis is already active as following spindle/axis in another coupling.

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program the spindle speed under address S in [rpm] (in spite of axis mode); the direction of rotation is indicated 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.

16762 Channel %1 block %2 spindle %3 thread function is active

Parameters: %1 = channel number
%2 = block number, label
%3 = spindle number

Definitions: Programming error:
The spindle function cannot be executed at the moment.
This alarm occurs when the spindle is linked with the axes by an interpolation function.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 2007 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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: A function has been programmed for an axis that requires a measuring system. According to MD 30 200 NUM_ENCS, this machine axis does not have a measuring system.

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

Remedy: Remove the relevant function (e.g. SPOS) from the part program or enter an existing measuring system in MD 30 200 NUM_ENCS.

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.

Reaction: Alarm display.

Remedy: Set the "Enable following axis overlay" VDI signal.

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

16772 Channel %1 block %2 axis %3 is the slave axis, the coupling is being opened

Parameters: %1 = channel number
%2 = block number, label
%3 = axis, spindle

Definitions: The axis is active as a following axis in a coupling. The coupling is opened in REF mode. The alarm can be suppressed with machine data setting 11410 SUPPRESS_ALARM_MASK Bit29 = 1.

Reaction: Alarm display.

Remedy: The coupling is closed again when REF mode is ended.

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

16773 Channel %1 axis %2 is the following axis. The axis/spindle disables of leading axes %3 and %4 differ from one another.

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = Axis, spindle
%4 = Axis, spindle

" Siemens AG 2007 All Rights Reserved

Definitions: The axis is active in a coupling as a following axis. The master axes have different states regarding axis/spindle disable.
The alarm can be suppressed with machine data 11415 SUPPRESS_ALARM_MASK_2 Bit0 =1.

Reaction: Alarm display.

Remedy: Set the same axis/spindle disable for all master axes.

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

16774 Channel %1 Synchronization aborted for slave axis/spindle %2

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

Definitions: For the indicated axis, the synchronization procedure (EGONSYN or EGONSYNE) was aborted.
There are several reasons for aborting the synchronization process:
- RESET
- End of program
- Axis goes to follow-up mode
- Rapid stop caused by an alarm

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

Remedy: If the abort of the synchronization procedure can be tolerated or is intended, the alarm can be suppressed with machine data 11410 SUPPRESS_ALARM_MASK Bit31 = 1.
Only applicable for electronic gear (EG):
If it is not possible to abort the synchronization procedure, you can achieve it by specifying the block change criterion FINE in EGONSYN or EGONSYNE.

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

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. One possible cause:
The spindle/axis has been under the control of the PLC and has not yet been released.

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

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 will create a ring coupling when other couplings are taken into account. This ring coupling cannot be uniquely computed.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Please inform the authorized personnel/service department. Configure the coupling accordingly in MD or correct the NC part program (channel MD: COUPLE_AXIS_n).

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 2007 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

16786 Channel %1 block %2 coupling to master spindle %3 already exists

Parameters: %1 = channel number
%2 = block number, label
%3 = master spindle number

Definitions: An attempt has been made to activate a coupling with a slave spindle that is already configured in an active coupling with another master spindle. The number of master spindles is restricted to one with the synchronous spindle function. The existing active master spindle is displayed as the last alarm parameter.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: The existing coupling must be separated before the new coupling is activated. The ELG function must be used if a coupling with several master spindles / axes is required.

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.

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

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Configure link accordingly in MD (channel MD: 42300 COUPLE_RATIO_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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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 vocabulary word 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)

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.
In the displayed NC block, replace the vocabulary word DC 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:
MD 30310 ROT_IS_MODULO
MD 30320 DISPLAY_IS_MODULO

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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

" Siemens AG 2007 All Rights Reserved

Definitions: The vocabulary word ACP (Absolute Coordinate Positive) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.
In the displayed NC block, replace the vocabulary word ACP 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 with modulo conversion by means of the axis-specific MD 30300 IS_ROT_AX and MD 30310 ROT_IS_MODULO.
Corresponding machine data:
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 vocabulary word ACN (Absolute Coordinate Negative) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

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

Remedy: Please inform the authorized personnel/service department.
In the displayed NC block, replace the vocabulary word ACN 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 with modulo conversion by means of the axis-specific MD 30300 IS_ROT_AX and MD 30310 ROT_IS_MODULO.
Corresponding machine data:
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 outside the range of 0 - 359.999 was programmed for a modulo axis.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program a position within the range from 0 - 359.999.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16903 Channel %1 program control: action %2<ALNX> not allowed in the current state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The relevant action cannot be processed now. This can occur, for example, when machine data are read in.

Reaction: Alarm display.

Remedy: Wait until the previous process is finished or abort with Reset and repeat the operation.

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

" Siemens AG 2007 All Rights Reserved

16904 Channel %1 program control: action %2<ALNX> not allowed in the current state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The operation (program, JOG, block search, reference point, etc.) cannot be started or continued in the current status.

Reaction: 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<ALNX> not allowed

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: Operation cannot be started or continued. A start is only accepted when an NC 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.

Reaction: Alarm reaction in Automatic mode.

Remedy: Check the program and channel status.

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

16906 Channel %1 program control: action %2<ALNX> is aborted due to an alarm

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The action was aborted due to an alarm.
Operation cannot be started or continued. A start is only accepted when an NC function can be started.

Reaction: 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<ALNX> only possible in stop state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: This action may only be performed in Stop state.

Reaction: Alarm display.

Remedy: Check the program and channel status.

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

16908 Channel %1 action %2<ALNX> only possible in reset state or at the block end

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The action %2 may only be performed in Reset state or at end of block.

Reaction: Alarm display.

Remedy: Check the program and channel status.

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

" Siemens AG 2007 All Rights Reserved

16909 Channel %1 action %2<ALNX> not allowed in current mode

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: You have to activate a different operating mode for the activated function.

Reaction: 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.

Reaction: Alarm display.

Remedy: Once overstoring has been 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<ALNX> only possible in reset state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The action can be performed only in the Reset state.
Example: Program selection through HMI or channel communication (INIT) can only be performed in Reset state.

Reaction: 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<ALNX> not allowed

Parameters: %1 = channel number
%2 = mode group number
%3 = action number/action name (see Section 1.5 Action List)

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!

Reaction: Alarm display.

Remedy: Either activate the RESET key to reset program processing, or select the mode in which the program was being processed previously.

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

16914 Mode group %1 channel %2 mode change: action %3<ALNX> not allowed

Parameters: %1 = channel number
%2 = mode group number
%3 = action number/action name (see Section 1.5 Action List)

Definitions: Incorrect mode change, e.g.: AUTO (MDAREF

Reaction: Alarm display.

Remedy: Check operation or selected operating mode.

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

" Siemens AG 2007 All Rights Reserved

16915 Channel %1 action %2<ALNX> not allowed in the current block

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

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.

Reaction: 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<ALNX> not allowed in the current state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: Block processing cannot be repositioned at the moment. It is also not possible to switch operating modes.
The 2nd parameter describes which action should be used to reposition the block processing.

Reaction: Alarm display.

Remedy: Let the program continue to a repositionable NC block or modify part program.

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

16919 Channel %1 action %2<ALNX> is not allowed due to a pending alarm

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: This action cannot be performed due to an alarm, or the channel is in an error state.

Reaction: Alarm display.

Remedy: Press the RESET key.

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

16920 Channel %1 action %2<ALNX> is already active

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: An identical action is still in progress.

Reaction: Alarm display.

Remedy: Wait until the previous action is finished and then repeat the operation.

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

16922 Channel %1 subprograms: action %2<ALNX> maximum nesting depth exceeded

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: Various actions can cause an interruption in the current processing operation.
Depending on the action, internal routines are activated. These programs can be interrupted in the same manner as the NC program. Unlimited nesting depth is not possible for internal routines due to memory limitations.
Example:
An interrupt interrupts the current program run. Other interrupts with higher priorities interrupt the previously active internal program runs.
Examples of actions are dry run, decoding single block, delete distance to go, etc.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Press the RESET key.
As a first step, check the program nesting depth and reduce or eliminate interruption.
Example:
The approach block in a repositioning process should not be interrupted repeatedly.

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

16923 Channel %1 program control: action %2<ALNX> not allowed in the current state

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The current processing run cannot be stopped because a preprocessing run is active.
This applies to, for example, loading machine data and block searches until the search object is found.

Reaction: Interface signals are set.
Alarm display.

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 are changed during program testing. The data cannot be correctly automatically when the program test is complete.
This error message prompts the user to make a backup copy of the data or to reimport the data after the operation is terminated.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
Save tool data to HMI and reimport data after "ProgtestOff".

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

16925 Channel %1 program control: action %2<ALNX> not allowed in the current state, action %3<ALNX> active

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: The action has been refused since a mode or sub-mode change (change to automatic mode, MDA, JOG) is currently in progress.
Example:
This alarm message is displayed if the Start key is pressed while a mode or submode changeover (e.g. from AUTOMATIC to MDA) is in progress and the NC has not yet confirmed the mode selection.

Reaction: 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 = Aktion
%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)

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

Remedy: Repeat action.

" Siemens AG 2007 All Rights Reserved

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

16927 Channel %1 action %2<ALNX> at active interrupt treatment not allowed

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: This action may not be activated during interrupt processing (e.g. mode change).

Reaction: 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<ALNX> not possible

Parameters: %1 = channel ID
%2 = action number/action name (see Section 1.5 Action List)

Definitions: A program interrupt has been activated in a non REORG capable block.
Examples of possible program interrupts in this case:
- Travel to fixed stop
- VDI channel delete distance to go
- VDI axial delete distance to go
- Measuring
- Software limit
- Axis replacement
- Axis is exited follow-up mode
- Servo disable
- Gear stage change with actual GS does not match setpoint GS

The relevant block is a:
a collection block for block search (except for last collection block) the Overstore Off block.

Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
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 function MSG must be packed into separate NC blocks. In order to prevent dips in velocity, these blocks are appended to the following NC block by an internal NC function (with WAITMC to the preceding 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, an auxiliary function, Stopre, dwell time, etc.

Reaction: Correction block is reorganized.
Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Program an executable NC block between the previous and the current NC block.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

16931 Channel %1 subprograms: action %2<ALNX> maximum nesting depth exceeded

Parameters: %1 = channel number
%2 = action number/action name (see Section 1.5 Action List)

Definitions: Various actions can cause an interruption in the current processing operation.
Depending on the action, internal routines are activated. These can be interrupted in the same manner as the user program. Unlimited nesting depth is not possible for internal routines 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.
Examples of actions are mode change, Slash On/Off

Reaction: Interface signals are set.
Alarm display.
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 work offset or base frame) which is also written by the part program 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 work offset

Reaction: 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.

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

Remedy: Abort program with Reset.

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 (see Section 1.5 Action List)

Definitions: Reorg events are, e.g. abort subprogram, delete distance-to-go and interrupts, axis replacement, exit follow-up state.

" Siemens AG 2007 All Rights Reserved

Two Reorg events overlap in this situation. In this case, the 2nd Reorg event affects the 1st block that has been generated by the preceding event, (e.g. axis replacement is forced twice in quick succession). Axis replacement leads to Reorg in the channels from 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 interpreter stop or by decode single block.

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

Remedy: Abort program 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.

Reaction: 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.

Reaction: Alarm display.

Remedy: Abort program with Reset.

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.

Reaction: 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 (see Section 1.5 Action List)

Definitions: Typical examples of Reorg events are abort subprogram, delete distance-to-go and exit from follow-up state.
These events wait for the end of a gear change. However, the maximum waiting period has elapsed.

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

Remedy: Abort program with Reset.

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

" Siemens AG 2007 All Rights Reserved

16939 Channel %1 action %2<ALNX> rejected due to active gear change

Parameters: %1 = channel ID
%2 = action number/action name (see Section 1.5 Action List)

Definitions: Reorganization events that are possible in Stop state, e.g. mode change, wait for the end of the gear change. However, the maximum waiting period has elapsed.

Reaction: Interface signals are set.
Alarm display.

Remedy: Repeat action.

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 (see Section 1.5 Action List)

Definitions: Reorganization events wait for the end of a gear change. The alarm is displayed during the waiting period.

Reaction: Alarm display.
Warning display.

Remedy: This is a self-clearing message.

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

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 20108 PROG_EVENT_MASK forces an asynchronous subprogram to be triggered automatically on RESET or Power On. 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:
The asynchronous subprogram does not exist
(/ N CMA DIR/ _N_PROG_EVENT_SPF)
No READY signal (due to alarm)

Reaction: Alarm display.

Remedy: Load program
Check MD 11602 ASUP_START_MASK.
Acknowledge the alarm.

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.

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

Remedy: Alarm can be switched off with $MN_SERUPRO_MASK bit 1.

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

" Siemens AG 2007 All Rights Reserved

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
- Axis replacement
- Delete distance-to-go
- Measuring

Reaction: 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.

Reaction: 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 cannot become active until 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 traversal at high speed does not start before the next block.

Reaction: Alarm display.

Remedy: Alarm can be switched off via MD 11410 SUPPRESS_ALARM_MASK Bit17==1.

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

" Siemens AG 2007 All Rights Reserved

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 programm command and $MC_DISABLE_PLC_START==FALSE was set.

Reaction: 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.

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.

Reaction: 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.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

==========
Example
Ch 3Ch 5Ch 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.
==========

Reaction: Alarm display.

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 3Ch 5Ch 7
WAITM(99,3,5,7) WAITM(99,3,5,7) WAITM(99,3,5,7)
==========
Sample solution B:
Ch 3Ch 5Ch 7
WAITM(99,3,5) WAITM(99,3,5)
WAITM(88,5,7) WAITM(88,5,7)
==========
Sample solution C:
Ch 3Ch 5Ch 7
WAITM(88,5,7) WAITM(88,5,7)
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: Info alarm
The block search was not performed on the interruption block, but stopped shortly beforehand on another block. This so-called "hold block" is generated by part program command IPTRLOCK, or defined implicitly in MD 22680 AUTO_IPTR_LOCK. The purpose of this function to prevent block searches from taking place in critical program areas (e.g. during hobbing operations). The alarm therefore indicates that the search has stopped on another block rather than on the interrupted block. This behavior is desired and the alarm serves only informational purposes.

Reaction: Alarm display.

Remedy: MD 11410 SUPPRESS_ALARM_MASK MD 22680 AUTO_IPTR_LOCK and NC 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: The part programmer can use NC commands IPTRLOCK and IPTRUNLOCK to identify a program section as "search-suppressed". Any search attempted in this program section is acknowledged with alarm 16951. In other words: Activation of this alarm indicates that the operator has started a search (type Serupro) for a search target located in a program area that must not be searched! This kind of program area can also be defined implicitly with machine data 22680 AUTO_IPTR_LOCK.
Note:
The alarm cannot be generated until the simulation during the block search is finished. The alarm cannot be displayed immediately a block search is started.

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

Remedy: MD 11410 SUPPRESS_ALARM_MASK, MD 22680 AUTO_IPTR_LOCK and NC command IPTRLOCK

" Siemens AG 2007 All Rights Reserved

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

16952 Channel %1 start program command not possible due to MDI

Parameters: %1 = channel ID

Definitions: The NCK is currently processing an ASUB in MDA mode. In this state, it is not legal to issue a "Start" program command for another channel. Notice! If an asynchronous subprogram is started from JOG, the NCK can switch internally to MDA if it was previously in this mode and has not been RESET. Note: With-out this alarm, the MDA buffer of the other channel would always be started.

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

Remedy: Start asynchronous subroutine in AUTO or AUTO->JOG

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

16953 Channel %1 For slave axis %2 SERUPRO not allowed, as master axis %3 not subject to axis/spindle disable

Parameters: %1 = Channel number
%2 = Slave axis name, following spindle number
%3 = Master axis name, master spindle number

Definitions: Currently, the alarm occurs only in combination with the SERUPRO action. SERUPRO stands for search via program test.
SERUPRO is possible only with an active coupling, if the axis/spindle disable is active for all master axes/spindles of the slave axis/spindle

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

Remedy: Set axis/spindle disable of the master axis

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

16954 Channel %1 block %2 programmed stop prohibited in stop delay area

Parameters: %1 = channel ID
%2 = block number, label

Definitions: A program command which causes a stop has been used in a program area (stop-delay area) which is parenthesized by DELAYFSTON and DELAYFSTOF. Except for G4, no commands which cause even a brief stop may be used in such areas. A stop-delay area can also be defined via MD 11550 STOP_MODE_MASK.

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

Remedy: MD 11550 STOP_MODE_MASK and NC command DELAYFSTON DELAYFSTOF

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

16955 Channel %1 stop in stop delay area is delayed

Parameters: %1 = channel ID

Definitions: An event which causes a stop has been detected in a program area (stop-delay area) which is parenthesized by DELAYFSTON and DELAYFSTOF. The stop is delayed and executed after DELAYFSTOF. A stop-delay area can also be defined via MD 11550 STOP_MODE_MASK.

Reaction: Interface signals are set.
Alarm display.

Remedy: MD 11550 STOP_MODE_MASK and NC command DELAYFSTON DELAYFSTOF

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

" Siemens AG 2007 All Rights Reserved

16956 Channel %1 program %2 cannot be started due to global start disable.

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

Definitions: The program selected in this channel cannot be started as "Global start disable" had been set.

Note:
PI "_N_STRTLK" sets the "Global start disable" and PI "_N_STRTUL"
deletes the "Global start disable".
The alarm is switched on with $MN_ENABLE_ALARM_MASK bit 6.

Reaction: Alarm display.

Remedy: Delete the "Global start disable" and restart.

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

16957 Channel %1 Stop-Delay area is suppressed

Parameters: %1 = Channel ID

Definitions: The program area (Stop-Delay area), which is put into brackets through DELAYFSTON and DELAYFSTOF, could not be activated. Every stop therefore becomes effective immediately and is not delayed!
This occurs every time, when braking into a stop Stop-Delay area, i.e. a braking process starts before the Stop-Delay area and ends not earlier than in the Stop-Delay area.
If the Stop-Delay area is entered with override 0, the Stop-Delay area
can also not be activated (example: a G4 before the Stop-Delay area allows the user to reduce the override to 0 and the next block in the Stop-Delay area
then starts with override 0 and the alarm situation described occurs.)
$MN_ENABLE_ALARM_MASK Bit-7 switches on this alarm.

Reaction: Interface signals are set.
Alarm display.

Remedy: $MN_STOP_MODE_MASK and language command DELAYFSTON DELAYFSTOF

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

16959 Channel %1 action %2<ALNX> prohibited during simulation block search.

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

Definitions: The function (2nd parameter) must not be activated during simulation search.

Reaction: Alarm display.

Remedy: Wait for search end.

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

16960 Channel %1 action %2<ALNX> prohibited during EXECUTE PROGRAM AREA.

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

Definitions: The function (2nd parameter) must not be activated during EXECUTE PROGRAM AREA.

Reaction: Alarm display.

Remedy: Wait for end of program area EXECUTE.

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

" Siemens AG 2007 All Rights Reserved

16961 Channel %1 action %2<ALNX> prohibited during syntax check.

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

Definitions: The function (2nd parameter) must not be activated during the syntax check.
Comment: The syntax check is served by the following PI services:
_N_CHKSEL _N_CHKRUN _N_CHKABO

Reaction: Alarm display.

Remedy: Wait for the end of the syntax check, or
Cancel the syntax check with reset, or
Cancel the syntax check with PI _N_CHKABO.

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

16962 Channel %1 NCK computing time reduced, start is not allowed.

Parameters: %1 = Channel number

Definitions: The computing time available to the NCK has been reduced, starts have therefore been locked. The computer performance is inadequate for smooth program execution. The computing time of the NCK may have been reduced by the HMI because of an HMI part program simulation.

Reaction: Alarm display.

Remedy: Wait for the simulation to end or press RESET in any channel.

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

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.

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

Remedy: Please inform the authorized personnel/service department.
• Edit 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 quantity of tool data in the NC is limited.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Delete any unnecessary tools.

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. when creating tool offset memory).

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

Remedy: Make more memory available to the data management system for subroutine calls and tool offsets

" Siemens AG 2007 All Rights Reserved

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

17018 Channel %1 block %2 incorrect value for parameter %3

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

Definitions: An incorrect value has been assigned to the stated parameter.
Only the following values are permissible for the parameter $P_WORKAREA_CS_COORD_SYSTEM
=1 for workpiece coordinate system
=3 for settable zero system.

Reaction: Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Assign another value.

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 (e.g. arithmetic parameter) with invalid 1st array index.
e.g. R2000 = 5; parameter number 2000 not defined

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Use the machine axis name as the axis index.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

17050 Channel %1 block %2 illegal value

Parameters: %1 = channel number
%2 = block number, label

Definitions: A frame component other than TRANS, ROT, SCALE or MIRROR was addressed when an individual frame element was accessed.
Frame components are selected either by the vocabulary words
TRfor offset (TRANS, internal 0)
RTfor rotation (ROT, internal 1)
SCfor scaling (SCALE, internal 3) and
MIfor 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=$P_UIFR[2, X, RT] can also be programmed as:
R10=$P_UIFR[2, X, 1]

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

Remedy: Address frame components only with the vocabulary words 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.

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

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 allocated for one symbol has been exceeded.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 has been made to write a protected variable or MD for which the user has no access authorization.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Remove write access operations to protected variable from the NC program or machine data file.
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
Definitions: An attempt was made to write an MD with a value that is smaller than the defined lower limit.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Please inform the authorized personnel/service department.
Determine the input limits of the MD 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
Definitions: An attempt was made to write an MD with a value that is higher than the defined upper limit.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Please inform the authorized personnel/service department.
Determine the input limits of the MD 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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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 = no. of input
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.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
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

" Siemens AG 2007 All Rights Reserved

Definitions: An attempt was made to read or set a digital NCK output 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 = no. of input

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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).

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 = number

" Siemens AG 2007 All Rights Reserved

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 MD:
10360 FASTIO_DIG_NUM_OUTPUTS and
10310 FASTIO_ANA_NUM_OUTPUTS

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program fewer digital/analog outputs in one 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.

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 although no tool was selected beforehand.

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

Remedy: Program or activate a tool offset in the NC part program.
Example:
N100 G.. ... T5 D1 ... LF

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.

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

Remedy: - SYSTEM ERROR – Please inform the authorized personnel/service department.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.
All tool parameters defined?
The dimensions of the tool edge must have been entered previously either on the operator panel or via the RS-232-C interface.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

17181 Channel %1 block %2 T no.= %3, D no.= %4 not existing

Parameters: %1 = channel number
%2 = block number, label

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 HMI 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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

17183 Channel %1 block %2 H number already available in T no.= %3, D no.= %4

Parameters: %1 = channel number
%2 = block number, label
%3 = T number
%4 = D number

Definitions: Each H number (apart from H=0) may only be issued once in a TO unit. The cutting edge specified already has the H number. If the H number is to be issued several times, machine data 10890 must be set as bit 3 = 1.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify programs:
Select another H number

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 tool list of channel %1 is not unique.
The specified T numbers %3 and %4 each have an offset with number %2.

Reaction: Interface signals are set.
Alarm display.

Remedy: Ensure that the D numbers within the TO unit are unique.
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 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 on the operator panel or via the RS-232-C.

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 NC does not recognize was programmed.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 NC (via MMC) and then 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.
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.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Check access to tool:
Are the parameters of the NC 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.

" Siemens AG 2007 All Rights Reserved

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 involved in a machining operation. Tool data for tools involved in the current machining operation may not be deleted.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Deselect tool

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

17224 Channel %1 block %2 tool T/D= %3 - tool type %4 is not permitted

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Incorrect T no. / D no.
%4 = Incorrect tool type

Definitions: On this system, it is not possible to select tool offsets of the indicated tool types.
The variety of tool types can both be limited by the machine OEM and be reduced on individual control models.
Only use tools of the tool types permitted for this system.
Check whether an error has occurred on defining the tool.

Reaction: Correction block is reorganized.
Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Correct the NC program or correct the tool data

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Correct the NC program: Assign the value of the machine data or 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.

" Siemens AG 2007 All Rights Reserved

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 vocabulary words CIC, CAC or CDC that has not been defined as an indexing axis in the machine data.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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:
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).
MD 10900: INDEX_AX_LENGTH_POS_TAB_1
MD 10920: INDEX_AX_LENGTH_POS_TAB_2 (number of positions for 1st/2nd indexing axis)
Default: 0 Maximum value: 60
MD 10910: INDEX_AX_POS_TAB_1 [n]
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.

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

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

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.

Reaction: 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 necessary.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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 date 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)
Or, with equidistant distances, the programmed index is smaller or equal 0.
Or, an attempt is made with a MOV movement to travel to a position outside the permitted area.

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

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 programmed preset axis is involved in the current transformation. This means that is it not possible to set the actual value memory (preset) for this axis.
Example:
Machine axis A must be set to the new actual value A 100 at the absolute position A 300.
Example:
N100 G90 G00 A=300
N101 PRESETON A=100

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Avoid preset actual value memory for axes which are participating in a transformation or deselect the transformation with vocabulary word TRAFOOF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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 addressed by vocabulary word POS or POSA is involved in the active transformation. Therefore, it cannot traverse as a positioning axis.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Remove POS or POSA instruction from the part program block or deselect transformation with TRAFOOF beforehand.

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!

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Remove G75 instruction from the part program block or deselect transformation with TRAFOOF beforehand.

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!

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Remove G74 instruction, or the machine axes involved in transformation, from the part program block or deselect the transformation with TRAFOOF beforehand.

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 illegal.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: First switch off the transformation function.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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 when a transformation is active. It might be possible to program the function in a different coordinate system. The axis identifier is used to select the coordinate system.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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 vocabulary word 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].

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Program vocabulary word 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).

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

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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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 applies:
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 or number of contour element <2 or >MAXNUM_CONTOURNO_PROTECTAREA.
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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
If the alarm is output on ramp-up (2nd parameter: "INIT" instead of block number), "Channel not ready to operate" will be set.

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.
If the alarm occurs during the control ramp up, the system variables $SN_PA_... for the stated protection zone must be corrected. Then make a restart. If the incorrect date is not identifiable, the immediate activation of the protection zone can be removed and the system variables of the protection zone can be rewritten with the aid of NPROTDEF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.
If the alarm occurs during NC program execution, the current block can be changed. This way, the NPROT parameters can also be adjusted. However, if there is an error in the definition of the protection zone, the NC program must be aborted and the definition must be corrected under NPROTDEF.
If the alarm occurs on control ramp-up, system variables $SN_PA_... must be corrected for the specified protection zone. This can be done by downloading an Initial.ini file that includes the relevant corrected date. If afterwards a restart is performed again, the alarm will have been removed provided that the data are consistent.

" Siemens AG 2007 All Rights Reserved

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 or number of the contour element <2 or >MAXNUM_CONTOURNO_PROTECTAREA.
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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
If the alarm is output on ramp-up (2nd parameter: "INIT" instead of block number), "Channel not ready to operate" will be set.

Remedy: Please inform authorized personnel / the service department.
1. Reduce the number of simultaneously active protection zones (MD).
2. Modify part program:
- Delete other protection zones.
- Preprocessing stop.
When the alarm occurs on control ramp-up, system variables $SC_PA_... must be corrected for the specified protection zone. Afterwards perform a restart. If the erroneous data cannot be recognized, the protection zone's immediate activation can be removed and the system variables of the protection zone can be written again by means of CPROTDEF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.
The current block can be changed if the alarm occurs during NC program execution. The CPROT parameters can also be adjusted. However, if the error lies in the definition of the protection zone, the NC program must be aborted and the definition corrected under CPROTDEF.
If the alarm occurs on control power-up, the system variables $SC_PA_... must be corrected for the specified protection zone. This can be done by downloading an Initial.ini file that includes the relevant corrected data. If another restart is then made, the alarm will have been eliminated provided that the data are now consistent.

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.

Remedy: Modify part program.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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 travel to fixed stop"
1:"Select travel to fixed stop"

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

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.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with NC START or RESET key and continue the program.

18300 Channel %1 block %2 frame: fine shift not possible

Parameters: %1 = channel number
%2 = block number, label

Definitions: Allocation of a fine offset to settable frames or the basic frame is not possible since MD 18600 MM_FRAME_FINE_TRANS is not equal to 1.

Reaction: Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Modify program or set MD 18600 MM_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 global frames.

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

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.

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

Remedy: Modify part program.

" Siemens AG 2007 All Rights Reserved

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 configurd with G58 and G59.

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

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 permissible to change the geometry axis assignment because the current frame contains rotations.

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

Remedy: Change NC program or set other mode with MD 10602 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 is assigned to the channel axis.
Channel-specific frames cannot be programmed with machine axis names if no corresponding channel axis is configured for the machine axis.

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

Remedy: Modify part program.

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

18400 Channel %1 block %2 language change not possible:%3

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 might be specified (see parameter 3):
1. Error in machine data settings
2. Active transformation

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

Remedy: Remedy the specified cause of the error before selecting the language.

" Siemens AG 2007 All Rights Reserved

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 34 030 REFP_MAX_CAM_DIST (phase 1 of referencing). (This error occurs only with incremental encoders).

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

Remedy: Please inform the authorized personnel/service department.
There are 3 possible causes of error:
1.The value entered in MD 34 030 REFP_MAX_CAM_DIST is too small.
Calculate the maximum possible distance from the start of referencing to the reduction cam and compare with the setting in MD 34 030 REFP_MAX_ CAM_DIST, increase the MD if necessary.
2.The cam signal is not received by the PLC input module.
Actuate the reference point switch manually and check the input signal at the NC/PLC interface (section: 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

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).

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

Remedy: 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 a longer cam.
When the axis has stopped on the cam, a check must be made to see whether the signal ”DECELERATION REFERENCE POINT APPROACH" (V 380x1000.7) is still present at the interface to the NC.
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 mark of the incremental encoder is not within a defined section.
Phase 2 of reference point approach ends when the zero mark of the encoder has been detected after the rising/falling edge of the PLC interface signal "DECELERATION REFERENCE POINT APPROACH" (V 380x1000.7) has given the trigger start. The maximum distance between the trigger start and the zero mark that follows is defined in MD 34 060 REFP_MAX_MARKER_DIST.
The monitor prevents a zero mark signal from being overtraveled and the next being evaluated as reference point signal! (Faulty cam adjustment or excessive delay by the PLC user program).

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

" Siemens AG 2007 All Rights Reserved

Remedy: Check the cam adjustment and make sure that the distance is sufficient between the end of the cam and the zero mark signal that follows. The path must be greater than the axis can cover in the PLC cycle time.
Increase MD 34 060 REFP_MAX_MARKER_DIST, but do not select a value greater than the distance between the 2 zero marks as this might deactivate the monitor!

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

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 MD 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.

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

Remedy: Determine the distance between 2 odd reference marks (reference mark interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in MD 34 060 REFP_MAX_MARKER_DIST.
Check the reference track of the scale including the evaluation electronics.

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: With a distance-coded linear measuring system, 2 reference marks have not been found within the defined search section (axis-specific MD 34 060 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 search distance MD 34060 REFP_MAX_MARKER_DIST, within which the two reference marks are expected, is counted commencing at the start point.

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

Remedy: Please inform the authorized personnel/service department.
Determine the distance between 2 odd reference marks (reference mark interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in MD 34 060 REFP_MAX_MARKER_DIST.
Check the reference track of the scale including the evaluation electronics.

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 the specified axes (e.g.: referencing aborted due to: missing servo enable signal, measuring system switchover, direction key released, etc.).

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

" Siemens AG 2007 All Rights Reserved

Remedy: Check reason for abort:
Servo enable missing (V 380x0001.1)
Traversing key signal + or - missing (V 380x0004.6 and .7)
Feed override = 0
The axis-specific MD 34 110 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 approach 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: MD 34 040 REFP_VELO_SEARCH_MARKER
Velocity tolerance:MD 35 150 SPIND_DES_VELO_TOL

Reaction: 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 MD for the approach velocity MD 34040 REFP_VELO_SEARCH_MARKER and/or increase the MD for the velocity tolerance MD 35 150 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 MD 34200 ENC_REFP_MODE = 6!

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

Remedy: Please inform the authorized personnel/service department.
Modify referencing mode MD 34200 ENC_REFP_MODE or install and configure a second encoder

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

20008 Channel %1 axis %2 reference point approach requires second referenced measuring system

Parameters: %1 = channel number
%2 = axis name, spindle number

Definitions: The setting MD 34200 ENC_REFP_MODE = 6 does not apply to this control system.

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

Remedy: Change referencing mode MD 34200 ENC_REFP_MODE

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Alarm display.

Remedy: Decide whether the axis is to be traversed by means of the direction keys or via the handwheel. End handwheel mode, with axial deletion of distance to go (V 380x0002.2) if appropriate.

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

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.

Reaction: Alarm display.

Remedy: Decide whether the axis is to be traversed by means of the direction keys or via the handwheel.

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

20052 Channel %1 axis %2 already active

Parameters: %1 = channel number
%2 = axis name, spindle number

Definitions: The axis is to traverse as a machine axis in JOG mode via the direction keys on the machine control panel. However, this is not possible because:
it is already traversing as a geometry axis
(V32001000.6V32001000.7,
V32001004.6V32001004.7 or
V32001008.6V32001008.7)
it is already traversing as a machine axis
(V38000004.6V38000004.7) or
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.

Reaction: 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.

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.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Correct (add to) the list of indexing positions by means of the machine data
MD 10900: INDEX_AX_LENGTH_POS_TAB_1
MD 10910: INDEX_AX_POS_TAB_1
MD 10920: INDEX_AX_LENGTH_POS_TAB_2
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 be derived.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

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 softkeys "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.

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 feed of the spindle/axis from which the feed is to be derived is 0.

Reaction: 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 necessary.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
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.
Correct part program.

" Siemens AG 2007 All Rights Reserved

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 created cannot be executed.

Reaction: Alarm display.

Remedy: The spindle/axis from which the feed is to be derived refers to itself.

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

20059 Channel %1 axis %2 already active due to %3

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

Definitions: The axis (machine axis, geometry axis or orientation axis) is to be traversed in operation mode "Automatic&Jog" (see $MN_JOG_MODE_MASK) by using the direction keys or a handwheel. This is not possible, as (see parameter 3):
1. the axis is active as a rotating spindle
2. the axis is a PLC axis
3. the axis is active as an asynchronous reciprocating axis
4. the axis is active as a command axis
5. the axis is active as a slave axis
6. a frame applies for a rotated coordinate system and an axis involved in the required JOG movement of the geometry axis is not available for this
7. an axis container rotation is activated via NCU link
Note: This alarm identifies an axis not capable of JOG which received a JOG order. In this case, the NCK will not proceed according to "Internal JOG".

Reaction: Alarm display.

Remedy: Wait for the axis to traverse or abort with distance-to-go delete or RESET.

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

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 a 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).

Reaction: Alarm display.

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.

Reaction: Alarm display.

Remedy: Register the axis as an orientation axis.

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

" Siemens AG 2007 All Rights Reserved

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
1st GEO axis: V32001000.6 and .7
2nd GEO axis: V32001004.6 and .7
3rd GEO axis: V32001008.6 and .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.

Reaction: 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.

Reaction: Alarm display.

Remedy: Activate an orientation transformation.

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

20064 Channel %1 axis %2 selection of several axes with an active taper angle is not permitted.

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

Definitions: With an active taper angle, only one geometry axis at the time can be traversed in JOG mode by pressing traversing keys. Simultaneous traversing of a geometry axis as a machine axis is not permitted either.

Reaction: Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Starting the geometry axis only if traversing of the other geometry axis or machine axis completed.

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

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 a geometry axis in JOG mode with revolutional feed, but no master spindle has been defined from which the actual speed could be derived.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.

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 softkeys "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 2007 All Rights Reserved

20070 Channel %1 axis %2 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.

Reaction: 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 necessary.

20071 Channel %1 axis %2 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.

Reaction: Alarm display.

Remedy: Specify smaller target position.
• Deactivate working area limitation.
• Set working area limitation differently.

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

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.

Reaction: 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 necessary.

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.

Reaction: 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.

Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

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 necessary.

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.

Reaction: 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.

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

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

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

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 "-"

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

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 override has been started in automatic mode. If the axis identifier is missing in the alarm with active velocity override 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.

Reaction: 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 necessary.

20081 Channel %1 axis %2 braking position cannot be accepted as a new reversing position

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

Definitions: On changing the reciprocation reversal from external sources, the braking position
cannot be accepted as a new reversing position, since changing the reversal point via handwheel or JOG key is active.

Reaction: Alarm display.

Remedy: Deselect VDI signal "Change reversal point" and reselect it either
- with "Reciprocation reversal from external sources" or
- by changing the reversal point by means of handwheel or
- by changing the reversal point via JOG key.

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

" Siemens AG 2007 All Rights Reserved

20082 Channel %1 block %2 coordinate system-specific working area limit %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"

Definitions: The displayed axis is operated as a "concurrent positioning axis", and the corresponding active coordinate system-specific working area limitation for the axis is violated. No traversing movement.
With an additional message to alarm 20140, the axis is traversed as a command axis.

Reaction: Alarm display.

Remedy: - Specify smaller target position.
- Deactivate working area limitation.
- Set working area limitation differently.
- Retract axis with JOG.

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

20083 Channel %1 axis %2 programmed position lies behind the coordinate system-specific working area limit %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"

Definitions: The axis is traversed as a reciprocating axis, and the target position (reversal position or end position) is located behind the corresponding, valid, coordinate system-specific working area limitation. The axis is not traversed.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

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

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.

Reaction: Alarm display.

Remedy: Turn the contour handwheel in the opposite direction. Program continuation: Alarm display disappears with alarm cause. No further operator action necessary.

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

20090 Axis %1 travel to fixed stop not possible. Check programming and axis data.

Parameters: %1 = axis name, spindle number

Definitions: The "Travel to fixed stop" function has been programmed with FXS[AX]=1 but the axis does not (yet) support this. Check MD 37000 FIXED_STOP_MODE. The function is not available for simulated axes.
On selection, no movement was programmed for axis AX. AX is a machine axis identifier.
It is always necessary to program a traversing movement in the selection block for the axis/spindle for which the "Travel to fixed stop" function is activated.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

Remedy: Please inform the authorized personnel/service department.
Check axis type
Check MD 37000
Is a machine axis movement missing in the approach block?

Program Continuation: Teileprogramm neu starten.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 travel to a fixed stop, the programmed end position has been reached or the traversing movement has been aborted. The alarm can be con-cealed by means of the machine data 37050 FIXED_STOP_ALARM_MASK.

Reaction: 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: Correct the part program and the settings:
Has the traversing block been aborted?
Should the axis position correspond to the programmed end position, then correct the end position.
If the programmed end position is in the workpiece, the trigger criterion must be checked.
Was the contour deviation that triggered the alarm too large? Has the torque limit been set too high?

Program Continuation: Teileprogramm neu starten.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 travel to fixed stop function is still selected.

Reaction: 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: Please inform the authorized personnel/service department.
Check the following possibilities:
Has the axis at the fixed stop also been moved by a traversing movement of geometry axes?
If the function selected even though the axis is stationary at the stop?
Has the function deselection been interrupted by a RESET?
Has the PLC switched the acknowledgement signals?

Program Continuation: Teileprogramm neu starten.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 outside the zero speed window ever since the function was selected.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

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 moved?
Position window for zero speed monitoring too narrow
(MD 37020 FIXED_STOP_WINDOW_DEF)
(SD 43520 FIXED_STOP_WINDOW).
Default is 1 mm in each case.

Program Continuation: Teileprogramm neu starten.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.
Possible reasons:
Because a pulse disable has occurred, the torque can no longer be produced.
The PLC has removed the acknowledgments.
The alarm can be re-configured in MD 11412 ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: 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: Has a pulse inhibit been issued by the infeed/regenerative feedback unit or the PLC? Have the acknowledgement bits have been reset by the PLC even though NCK has not requested deselection?

Program Continuation: Teileprogramm neu starten.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

Definitions: The current holding torque, when brake test selected, cannot be attained with the present parameterization of the brake test.

Reaction: 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_SAFE_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.

Reaction: Interface signals are set.
Alarm display.

Remedy: Note the supplementary conditions of the brake test, see additional info.

" Siemens AG 2007 All Rights Reserved

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

20097 Axis %1 incorrect travel direction brake test

Parameters: %1 = Axis name, spindle number

Definitions: Due to the selected travel direction, the brake test for the current load torque is performed with an incorrect torque.

Reaction: Alarm display.

Remedy:
- Perform the brake test for the other travel direction
- Adjust drive MD 1192 better to the current weight ratio. The alarm will occur only if the current torque deviates from MD 1192 by more than 5% when the brake is released.
- Activate the automatic determination of the load torque at the beginning of the brake test via MD $MA_SAFE_BRAKETEST_CONTROL, Bit 0 = 1.

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

20100 Channel %1: invalid configuration for digitizing

Parameters: %1 = Channel number

Definitions:
- The digitizing function expects the definition of 3 geometry axes in the channel.
- At the available baud rate for a transmission of the actual positions and setpoint velocities between the NC and the digitizing device, the interpolation cycle must be set to a minimum of 5ms.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform authorized personnel / the service department.
- Define 3 geometry axes for the digitizing channel by means of machine data.
- Use an interpolation cycle greater than 5ms.

Program Continuation: Switch control OFF - ON.

20101 Timeout during initialization of communication with the digitizer

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.

Reaction: Interface signals are set.
Alarm display.

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.

Reaction: Interface signals are set.
Alarm display.

Remedy:
- Before digitizing, activate a permitted transformation.
- Select 3-axis mode for 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 axes 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.

Reaction: Interface signals are set.
Alarm display.

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.

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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. 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.

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

Remedy: -

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

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.

Reaction: 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. 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.).

Reaction: 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 connection cable leading to the digitizing unit: In particular screening.

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

" Siemens AG 2007 All Rights Reserved

20139 Channel %1 block %2 motion-synchronous action: invalid marker

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

Definitions: Setting or deleting of a marker in the motion-synchronous action is not possible.
Possible causes:
SETM(): Maximum number of markers exceeded; marker has already been set.
CLEARM(): Specified marker is not within permissible value range.

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

Remedy: SETM(): use marker in valid value range; do not set the marker again.
CLEARM(): use marker in valid value range.

Program Continuation: Clear alarm with the RESET key. 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.

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

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).

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

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

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.

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

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.

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

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).

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

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.

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

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

" Siemens AG 2007 All Rights Reserved

- Programming error (see alarm 21701)

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

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.

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

Remedy: Change synchronous action.

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

20149 Channel %1 block %2 motion synchronous action: illegal index

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

Definitions: An invalid index was used for accessing a variable in the 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.
PROFIBUS I/O:
An invalid slot / I/O area index was used while reading/writing data.
Cause:
1.: Slot / I/O area index >= max. number of available slot / I/O areas.
2.: Slot / I/O area index references a slot / I/O area that has not been configured.
3.: Slot / I/O area index references a slot / I/O area that has not been released for a system variable.

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

Remedy: Use a valid index.

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.

Reaction: 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 necessary.

" Siemens AG 2007 All Rights Reserved

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

Reaction: 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 necessary.

20300 Channel %1 axis %2 orientation not possible

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

Definitions: On traversing the displayed (virtual) orientation axis, a tool orientation is to be set for which the kinematics of this machine are not possible.

Reaction: Alarm display.

Remedy: Abort the JOG movement and specify another (possible) change of orientation.

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

21550 Channel %1 axis %2 Travel from hardware limit switch not possible. Reason: %3

Parameters: %1 = channel number
%2 = axis name
%3 = cause

Definitions: An attempt has been made to retract a coupled following axis or an output axis in a transformation via the leading axis or input axis in a transformation. This is not allowed in the current situation.
Possible causes:
1 Retraction direction not legal
2 Coupling not synchronized
3 Retraction not legal for the active coupling
4 Reserved
5 Retraction not legal for the active transformation

Reaction: NC Start disable in this channel.
Alarm display.

Remedy: Remedy of error cause:
1 Specify a different travel direction
2 Deactivate the coupling and move axis(axes) separately
3 Deactivate the coupling and move axis(axes) separately
4 Reserved
5 Deactivate the transformation and move axis(axes) separately

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

21600 Monitoring for ESR active

Definitions: -

Reaction: NC not ready.
Alarm display.
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 necessary.

" Siemens AG 2007 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 for the active encoder (axial interface signal V390x0000.2) specified in axis-specific MD 36 300 ENC_FREQ_LIMIT [n] (n ... encoder number 1 or 2) has been exceeded. The reference of the actual value to the mechanical slide position may have been lost. The alarm can be re-configured in MD 11412 ALARM_REACTION_CHAN_NOREADY (NC not ready).

Reaction: 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 MD 36300 ENC_FREQ_LIMIT [0].

Program Continuation: Teileprogramm neu starten.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.

Reaction: The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
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: enable reset, cause %3

Parameters: %1 = channel number
%2 = axis name, spindle number

Definitions: The interface signal "Servo enable" (V 380x0002.1) has been set to 0 for the displayed axis even though an axis from the geometry grouping was moving.
The axes entered in MD array 20050 AXCONF_GEOAX_ASSIGN_TAB count as axes belonging to the geometry grouping. Servo enable must be applied for all configured geometry axes, regardless of whether they are currently in motion or not.

Reaction: The NC switches to follow-up mode.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Check the interface signal "Servo enable" (V 380x0002.1, e.g. by reading the PLC status display in the DIAGNOSIS operating area). Trace back the signal to the sections in the PLC user program at which it is linked and set/reset.

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.

Reaction: Alarm display.

Remedy: -

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

" Siemens AG 2007 All Rights Reserved

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" has been set at the NC/PLC interface (V 380x1000.0 or.1).

Reaction: NC Start disable in this channel.
Alarm display.

Remedy: 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, MD 36100 POS_LIMIT_MINUS, MD 36130 POS_LIMIT_PLUS2 and MD 36120 POS_LIMIT_MINUS2 as well as the interface signal for selection of the 1st/2nd software limit switch
(V 380x1000.2 and .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 switched from traverse mode and to "Follow-up" mode, for instance, because the pulse enable for the drive has been reset.

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

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.

Reaction: Local alarm reaction.
Interface signals are set.
Alarm display.
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 specified curve passes through the pole or a forbidden area of the transformation.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

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 transformation remains active when RESET is applied).

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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

Definitions: The machine kinematics does not allow the specified motion. Possible causes of transformation-related error:
With TRANSMIT:
A (circular) area exists around the pole, where positioning is not possible. The area exists because the tool reference point cannot be traversed into the pole. The area is defined by
The machine data ($MC_TRANSMIT_BASE_TOOL..).
The active tool length compensation (see $TC_DP..).
The inclusion of the tool length compensation depends on the selected working plane (see G17,..).
The machine stops at the edge of the area where positioning is not possible.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Modify the incorrectly specified tool length compensation

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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.

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.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
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)

Reaction: 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 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 vocabulary word MEAS or MEAW is already deflected and has switched. The probe signal must be reset (neutral position of probe) before another measurement can be taken.
The axis display is of no significance at the present time but an axis-specific evaluation has been planned for later stages of development.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Check start position for measurement or check probe signals. Are the cables and connectors in good order?

" Siemens AG 2007 All Rights Reserved

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

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 measuring mode
Invalid probe
Invalid encoder
Invalid number of measuring 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).

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Correct 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 probe has not yet responded.

Reaction: Alarm display.

Remedy: Check the traversing movement in the measurement block.
Is it essential for the activated probe to have switched by the time the specified axis position is reached?
Are probe, cable, cable distributor, terminal connections O.K.?
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.

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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: - Check probe
- Check start position for measurement
- Check program

" Siemens AG 2007 All Rights Reserved

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].

Reaction: Alarm display.

Remedy: Do not program values with $A_OUTA[..] = x that are higher 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).

Reaction: 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: 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

Definitions: The number of programmed auxiliary functions exceeds the maximum permissible number. This alarm can occur in conjunction with motion-synchronous actions: The maximum permissible number of auxiliary functions must not be exceeded in motion block and motion-synchronous actions.

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

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 required

Definitions: This alarm is activated via MD 27880 PART_COUNTER, bit 1: The number of counted workpieces {$AC_ACTUAL_PARTS or $AC_SPECIAL_PARTS} is equal to or already higher than the programmed value for the number of workpieces required {$AC_REQUIRED_PARTS} The channel VDI signal "Workpiece required reached" is output at the same time. The value for the number of the counted workpieces $AC_ACTUAL_PARTS is reset while the value of $AC_SPECIAL_PARTS remains.

Reaction: NC not ready.
Interface signals are set.
Alarm display.

Remedy: No program interrupt. Clear alarm display.

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

" Siemens AG 2007 All Rights Reserved

22000 Channel %1 block %3 spindle %2 gear stage change 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".

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

Remedy: Before the S word which requires a gear stage change, change into the open-loop control mode of the spindle.
The spindle is switched to open-loop control mode with:
M03, M04, M05 or M41 ... M45 from axis or positioning mode
Interface signal "Gear is changed" (V 38032000.3) from oscillation mode

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

22005 Channel %1 spindle %2 selected gear stage %3 not installed

Parameters: %1 = Channel number
%2 = Spindle number
%3 = Gear stage

Definitions: The first gear stage data block is active. The required gear stage is not installed in the 1st gear stage data block. The number of installed gear stages is configured in machine data 35090 $MA_NUM_GEAR_STEPS.
Examples for the occurrence of the alarm with 3 gear stages installed (MD 35090 $MA_NUM_GEAR_STEPS = 3):
* ...DO M44 or DO 45 was programmed in synchronized action for the spindle concerned.
* ...DO M70 was programmed and machine data 35014 $MA_GEAR_STEP_USED_IN_AXISMODE was larger than 3.

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

Remedy: Modify part program: Only those valid gear stages can be entered which have also been installed according to machine data MA_NUM_GEAR_STEPS.
Limit M70 configuration (MD 35014 $MA_GEAR_STEP_USED_IN_AXISMODE) to MD 35090 MA_NUM_GEAR_STEPS.

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

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 gear stage requested by the NC.
Note: Wherever possible, the requested gear stage should always be engaged.

Reaction: Alarm display.

Remedy: Correct the PLC program.

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

" Siemens AG 2007 All Rights Reserved

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' and 'ProgramTest' 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 in the deselection block is not operating in speed control mode or not active as a following axis or in a transformation. Retro-execution of a gear stage change when the above mentioned functions are deselected can be avoided by resetting bit 2 of machine data 35035 SPIND_FUNCTION_MASK.

Reaction: Alarm display.

Remedy: Change deselection block or block search target block to speed control mode (M3, M4, M5, SBCOF).

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 stage change. The required gear stage change position has not been reached.

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

Remedy: Correct sequence in the PLC.

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

22022 Channel %1 block %2 spindle %3 gear stage %4 is expected for axis mode.

Parameters: %1 = channel number
%2 = block number, label
%3 = spindle

Definitions: A gear stage for spindle operation in axis mode is configured in machine data 35014 GEAR_STEP_USED_IN_AXISMODE. The NC checks that this gear stage is engaged when the spindle is switched to this gear stage. It does this by comparing the gear stage configured in MD 35014 with the actual gear stage returned by the PLC (VDI interface "Actual gear stage A to C", V38002000.0 … 2). If the gear stages are not identical, this alarm is generated. When the spindle switches to axis mode in response to a programmed M70 command, the NC automatically engages or requests the gear stage configured in MD 35014. If the gear stage in this MD is already active, no gear stage change is requested. M40 remains active in both cases.

Reaction: Interface signals are set.
Alarm display.

Remedy: Program M70 before axis mode. Note MD 20094.

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

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 measuring system position although reference is made to it.

Reaction: Alarm display.

Remedy: Correct NC part program. Establish zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.

" Siemens AG 2007 All Rights Reserved

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) has been programmed, but no spindle encoder defined.
On activation of position control, the spindle speed is greater than the limit speed of the measuring system.

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

Remedy: Spindle with no encoder attached: NC language elements which require encoder signals must not be used.
Spindle with encoder attached: Enter spindle encoder in MD 30200 NUM_ENCS.

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

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: During referencing, the spindle rotated through a distance greater than axis-specific MD 34060 REFP_MAX_MARKER_DIST without detecting a reference mark signal. The check is made during spindle positioning with SPOS or SPOSA if the spindle has not previously operated under speed control (S=...).

Reaction: 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 and correct MD 34060 REFP_MAX_MARKER_DIST. The value entered specifies the distance in [mm] or [degrees] between 2 zero marks.

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 specified spindle was programmed as a spindle or axis, but a positioning operation was still in progress from the previous block (with SPOSA ... spindle positioning beyond block boundaries).
Example:
N100 SPOSA [2] = 100
:
N125 S2 = 1000 M2 = 04; error if spindle S2 is still running
; from block N100!

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

Remedy: Before programming the spindle/axis again after the SPOSA instruction, you should use a WAITS command to cause the program to wait for the programmed spindle position.
Example:
N100 SPOSA [2] = 100
:
N125 WAITS (2)
N126 S2 = 1000 M2 = 04

" Siemens AG 2007 All Rights Reserved

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 with an absolute encoder, only the referencing mode MD 34200 ENC_REFP_MODE = 2 is supported! ENC_REFP_MODE = 6 is not supported by SPOS!

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

Remedy: Change setting in MD 34200 ENC_REFP_MODE, switch 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.

Reaction: 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 measuring system position although reference is made to it.

Reaction: Alarm display.

Remedy: Modify part program. Establish 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 (setpoint or actual-value coupling) or the programmed function requires position control.

Reaction: Alarm display.

Remedy: Activate position control, e.g. by programming SPCON.

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

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 mark search velocity is not reached.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Check active speed limitation.
Configure a lower zero mark search velocity MD 34040 REFP_VELO_SEARCH_MARKER. Check tolerance range for the actual velocity in MD 35150 SPIND_DES_VELO_TOL. Set a different referencing mode MD 34200 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 mark search velocity is too high. The encoder limit frequency is exceeded for the active measuring system.

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

Remedy: Configure a lower zero mark search velocity MD 34040 REFP_VELO_SEARCH_MARKER. Check encoder limit frequency configured in MD 36300 ENC_FREQ_LIMIT and MD 36302 ENC_FREQ_LIMIT_LOW. Set a different referencing mode
MD 34200 ENC_REFP_MODE=7.

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

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!

Reaction: 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 and correct the start-up and optimization data of the drive actuator in accordance with the Installation and Start-up Guide.
Increase the tolerance window in machine data 35150 SPIND_DES_VELO_TOL.

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

22101 Channel %1 block %3 spindle %2 maximum speed for encoder resynchronization exceeded

Parameters: %1 = channel number
%2 = axis name, spindle number
%3 = block number, label

Definitions: The limit frequency of the active encoder (axis-specific MD 36300 ENC_FREQ_LIMIT[0]) has been exceeded with function G33 (thread cutting with encoder), G95 (revolutional feedrate) or G96 (constant cutting rate). As a result, the spindle is no longer synchronized.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Check whether the encoder is activated by interface signal: position measuring system V380x0001.5, or whether the limit frequency for this encoder is correctly preset in MD 36300 ENC_FREQ_LIMIT[0].
Check the maximum spindle speed setting in axis-specific MD 35130 GEAR_STEP_MAX_VELO_LIMIT and reduce if necessary.
Set an upper spindle speed limitation (which must be lower than the maximum encoder limit frequency) with G26 S... in the preceding NC block.

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

22150 Channel %1 block %3 spindle %2 maximum speed for position control exceeded

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

Definitions: The maximum encoder speed was exceeded with SPCON. Position control is no longer possible. The NC reduces the setpoint speed with the above functions until the active encoder is able to measure again. The alarm is issued if the encoder still reports the fault.

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

Remedy: - Program speed limit with G26.
- Reduce the maximum speed in the appropriate machine data.

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

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.

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

Remedy: Provide an interlock in the PLC user program so that no axis stop can be initiated when tapping is active. If the tapping operation must 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.

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

Remedy: Check the axis-/spindle-specific stop signals (V 380x0004.3)

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

" Siemens AG 2007 All Rights Reserved

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 after the last thread block!

Reaction: 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 = axis name, spindle number

Definitions: The spindle speed is so high for a thread cutting operation with G33 that the maximum axis velocity has been exceeded as a result of the programmed lead.

Reaction: Alarm display.

Remedy: Program a lower spindle speed or a speed limitation with G26 S ... or reduce the spindle speed prior to the thread block using SD 43 220 SPIND_MAX_VELO_G26 or the spindle override function.

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 is dependent on:
- Programmed thread lead decrease.
- Thread length.

Reaction: 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.

Reaction: Alarm display.

Remedy: Adjust the setting in SD 42010 THREAD_RAMP_DISP.

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

22290 Channel %1 spindle operation for transformed spindle/axis %2 not possible (reason: error code %3).

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

" Siemens AG 2007 All Rights Reserved

Definitions: It is impermissible to start a spindle as long as it is used by a transformation. Reason: spindle usage in a transformation requires axis operation which must not be exited.
This alarm may have the following reasons:
- Error code 1 : M3, M4 or M5 per synchronized action;
- Error code 2 : M41 through M45 per synchronized action;
- Error code 3 : SPOS, M19 per synchronized action;
- Error code 11 : DBB30 spindle stop;
- Error code 12 : DBB30 spindle start clockwise rotation;
- Error code 13 : DBB30 spindle start counterclockwise rotation;
- Error code 14 : DBB30 spindle positioning.

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

Remedy: Resolve the conflict, for example by deactivating transformation prior to spindle start.

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

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.

Reaction: Interface signals are set.
Alarm display.

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).

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

Remedy: Use more realistic (smaller) Preset values.

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

22400 Channel %1 option 'contour handwheel' not set

Parameters: %1 = Channel number

Definitions: The function 'contour handwheel' was activated without the neccesary option.
If the alarm occurs
- on selection of the contour handwheel via the PLC, then the contour handwheel has to be deselected in order to continue with the program
- on account of programming FD=0, then the program can be corrected and continued with the compensation block and NCSTART.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Set option
- Cancel the activation of the function 'contour handwheel'
- Modify part program.

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

" Siemens AG 2007 All Rights Reserved

25000 Axis %1 hardware fault of active encoder

Parameters: %1 = axis name, spindle number

Definitions: The signals from the currently active actual position encoder (interface signal V 380x001.5) are missing, or they are not co-phasal, or they are showing a ground fault/short circuit.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found.

Program Continuation: Switch control OFF - ON.

25001 Axis %1 hardware fault of passive encoder

Parameters: %1 = axis name, spindle number

Definitions: The signals from the currently inactive actual position encoder are missing, or they are not co-phasal, or they are showing a ground fault/short-circuit.

Reaction: 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. Deactivate monitoring with the appropriate interface signal (V 380x001.5).

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 is signaling contamination (only in measuring systems with contamination signal).

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Check the measuring system in accordance with the instructions given by the measuring device manufacturer.

Program Continuation: Teileprogramm neu starten.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 is signaling contamination (only in measuring systems with contamination signal).

Reaction: 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.

" Siemens AG 2007 All Rights Reserved

25020 Axis %1 zero mark monitoring of active encoder

Parameters: %1 = axis name, spindle number

Definitions: The position encoder pulses between 2 zero mark pulses are counted (hardware function). It checks that the encoder always outputs the same number of pulses between two zero marks. As soon as a difference is registered in the 4 low-significance counter bits, an alarm is triggered!

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: 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:
Transmission path: Check the actual-value connector on the motor for correct contacting, check encoder cable for continuity and short circuits or ground faults (loose contact?).
Encoder pulses: Encoder power supply within the tolerance limits?
Evaluation electronics: Replace or reconfigure the drive module used.
Monitoring can be switched off by setting MD 36310 ENC_ZERO_MONITORING to 0.

Program Continuation: Teileprogramm neu starten.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 mark pulses are counted (hardware function). A check is made in the interpolation cycle grid (default setting 4 ms) as to whether the encoder always issues the same number of pulses between the zero marks. As soon as a difference is registered in the 4 least-significance counter bits, an alarm is triggered!

Reaction: 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, check encoder cable for continuity and for short circuits or ground faults (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

Definitions: This alarm only occurs with absolute encoders:
a. Warning notice of missing absolute encoder adjustment (on the SIMODRIVE 611D or with PROFIdrive drives), that is if $MA_ENC_REFP_STATE equals 0. In this case, fine error code 0 is returned.
b. If a zero mark monitoring has been activated on the SIMODRIVE 611D for the absolute encoder (see $MA_ENC_ZERO_MONITORING): In this case, the absolute position of the absolute encoder could not be read without error:

" Siemens AG 2007 All Rights Reserved

Breakdown of fine error codes:
(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
This alarm is only displayed, as the absolute position itself is not required at this time for control/contour.
A frequent occurrance of this alarm indicates that the absolute encoder transfer or the absolute encoder itself is faulty, and that an incorrect absolute value could be determined in one of the next encoder selection or power on situations.

Reaction: Alarm display.

Remedy: a. Verify encoder adjustment (machine reference) or readjust encoder.
b. 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: The actual velocity of the axis is checked once in every IPO cycle. If there are no errors, the actual velocity can never exceed the setting 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.

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

Remedy: Check speed setpoint cable (bus cable).
Check the actual values and direction of position control.
Reverse the position control direction accelerates in an uncontrolled manner (axis-specific MD 32110 ENC_FEEDBACK_POL = < -1, 0, 1 >.
Increase monitoring limit in MD 36200 AX_VELO_LIMIT.

Program Continuation: Teileprogramm neu starten.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 --

Reaction: Alarm display.

Remedy: -

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

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. The monitor is activated after an adjustable axial time period in MD 36040
STANDSTILL_DELAY_TIME which follows completion of interpolation. A continuous check is made to ascertain whether the axis is within the tolerance threshold in MD 36030 STANDSTILL_POS_TOL.

" Siemens AG 2007 All Rights Reserved

The following scenarios are possible:
1.The interface signal SERVO ENABLE (V 380x0002.1) is zero because the axis is clamped mechanically. Due to mechanical influences (e.g. high machining pressure), the axis is pushed away from the permissible position tolerance.
2.With a closed position control loop (without clamping) – interface signal SERVO ENABLE(V 380x0002.1) is ”1” – the axis is pushed out of
position by high mechanical forces with low gain in the position control loop.

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

Remedy: Check MD 36040 STANDSTILL_DELAY_TIME and
MD 36030 STANDSTILL_POS_TOL and increase if necessary.
Estimate the machining forces and lower if necessary by feedrate reduction/speed increase.
Increase the clamping pressure.
Increase the gain in the position control loop by improved optimization (servo gain factor MD 32200 POSCTRL_GAIN).

Program Continuation: Teileprogramm neu starten.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.

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

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: Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group.
Restart part program.

25050 Axis %1 contour monitoring

Parameters: %1 = axis name, spindle number

Definitions: The NC 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.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Check tolerance value in MD 36400 CONTOUR_TOL, increase setting if necessary.
Check optimization of the position controller (servo gain factor in MD 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.
Check acceleration in MD 32300 MAX_AX_ACCEL. The position control is opened if the axis reaches current limits as a result of excessive acceleration. The "lost" actual value is "caught up" in the form of an overshoot once the control loop is closed again.
Improvement of speed controller optimization.
Check the mechanics (smooth running, inertial masses).

Program Continuation: Teileprogramm neu starten.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. A setting of 100% corresponds to the rated speed of the motor and therefore to rapid traverse velocity.
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. During this period, the setpoint is limited to the set maximum value (MD 36210 CTRLOUT_LIMIT).

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

Remedy: If the drive controller has been set correctly and if the machining conditions are those that normally prevail, then this alarm should not occur.
Check actual values: Local sluggishness of slide, speed dip and torque surge on workpiece/tool contact, travel to fixed obstacle, etc.
Check direction of position control: Does the axis accelerate uncontrollably?
Check the speed setpoint cable.

Program Continuation: Teileprogramm neu starten.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: Applies only to 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 MD 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 MD 36720 DRIFT_VALUE. This is not included in the drift monitoring because it acts like a voltage zero offset.

Reaction: Alarm display.

Remedy: 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 (temperature rise effects).

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

" Siemens AG 2007 All Rights Reserved

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 axis-specific MD 36 020 POSITIONING_TIME.
Exact stop coarse: MD 36000 STOP_LIMIT_COARSE
Exact stop fine: MD 36010 STOP_LIMIT_FINE

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

Remedy: 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; set gain as high as possible.
Check setting of servo gain factor (MD 32200 POSCTRL_GAIN) and increase if necessary.

Program Continuation: Teileprogramm neu starten.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!

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

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

" Siemens AG 2007 All Rights Reserved

25105 Axis %1 measuring systems differ considerably

Parameters: %1 = axis name, spindle number

Definitions: The two measuring systems are drifting apart, 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 36510 ENC_DIFF_TOL. This can only occur when both measuring systems are active (MD 30200 NUM_ENCS = 2) and referenced. The alarm can be re-configured in MD 11412 ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Please inform the authorized personnel/service department. Check machine data of active and selected encoders. Check machine data for encoder tolerance(MD 36510 ENC_DIFF_TOL).

Program Continuation: Teileprogramm neu starten.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.

Reaction: 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 one 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 position control. The number of this parameter set is outside the permissible limit (8 parameter sets: 0 ... 7 available).

Reaction: 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 the axis-/spindle-specific interface signals (V380x4001.0 - .2 "Select drive parameter set A, B, C").
One parameter set includes the following machine data:
MD 31050 DRIVE_AX_RATIO_DENOM [n]
MD 31060 DRIVE_AX_RATIO_NUMERA [n]
MD 32200 POSCTRL_GAIN [n]
MD 32810 EQUIV_SPEEDCTRL_TIME [n]
MD 32910 DYN_MATCH_TIME [n]
MD 36200 AX_VELO_LIMIT [n]

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

" Siemens AG 2007 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 identified 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.

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

Remedy: Evaluation of the drive alarms listed above.

Program Continuation: Teileprogramm neu starten.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: Group fault drive (self-clearing)

Reaction: Interface signals are set.
Alarm display.

Remedy: Wait for the drive.
This alarm reveals similar problems to alarm 25201 (see this alarm).
The alarm is active continuously during power-up if the drive is not communicating (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 necessary.

26000 Axis %1 clamping monitoring

Parameters: %1 = axis name, spindle number

Definitions: The clamped axis has been pushed out of its setpoint position. The permissible difference is defined in the axis-specific MD 36050 CLAMP_POS_TOL.
Axis clamping is activated with axis-specific interface signal
V 380x0002.3: "Clamping in progress".

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

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: Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group.
Restart part program.

" Siemens AG 2007 All Rights Reserved

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).

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

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: Teileprogramm neu starten.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: Rotary measuring system (MD 31000 ENC_IS_LINEAR[] == FALSE)
The number of encoder marks set in MD 31020 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.
Absolute measuring system with EnDat interface
(MD 30240 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 (MD 31000 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 (MD 31000 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)

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

Remedy: Please inform the authorized personnel/service department.
Compare 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.

" Siemens AG 2007 All Rights Reserved

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 MD 31030 LEADSCREW_PITCH is zero.

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

Remedy: Determine pitch of ballscrew (data from machine manufacturer or pitch measurement with spindle cover removed) and enter in MD 31030 LEADSCREW_PITCH (usually 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.

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

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 MD 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 MD 32250 RATED_OUTVAL or in MD 32260 RATED_VELO is zero.

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

Remedy: The rated motor speed should be entered in MD 32260 RATED_VELO.

Program Continuation: Teileprogramm neu starten.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

" Siemens AG 2007 All Rights Reserved

Definitions: Not all encoder types and/or output types are available in the current software status.
MD 30240 ENC_TYPE= 0Simulation
= 1Signal generator
= 2Rectangular encoder
= 4EnDat absolute encoder
MD 30130 CTRLOUT_TYPE= 0Simulation
= 1Standard

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

Remedy: Check and correct in MD 32240 ENC_TYPE and/or MD 30130 CTRL_OUT_TYPE.

Program Continuation: Switch control OFF - ON.

26014 Axis %1 machine data %2 invalid value

Parameters: %1 = axis name, spindle number
%2 = string: MD identifier

Definitions: MD contains an invalid value.

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

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: MD contains an invalid value.

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

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 contains an invalid value.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Repeat entry with correct value and then Reset.

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

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 contains an invalid value.

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

Remedy: Repeat entry with correct value and then Reset.

Program Continuation: Teileprogramm neu starten.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 MD 30110 CTRLOUT_MODULE_NR contains the same value for different axes.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.
Avoid duplicate setpoint assignment by correcting MD 30110 CTRLOUT_MODULE_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 13100 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.

Reaction: Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: If possible, modify the measuring motion in such a manner 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 13100 DRIVE_DIAGNOSIS[8].

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

" Siemens AG 2007 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 or while accessing encoder (refer to additional information for absolute encoder interface from error fine coding).

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Please inform the authorized personnel/service department.
Rectify hardware error, replace encoder if necessary.

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 is made without the encoder hardware (simulated encoder).

Reaction: Local alarm reaction.
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 possible, modify the measuring motion in such a manner 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 30240 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: The machine data contains an invalid value. This has therefore been changed by the software.

Reaction: 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: The machine data contains an invalid value. It has therefore been changed internally by the software to a valid value.

Reaction: Alarm display.

Remedy: Check MD.

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

" Siemens AG 2007 All Rights Reserved

26026 Axis %1 SINAMICS drive parameter P2038 value is not allowed.

Parameters: %1 = Axis name, spindle number

Definitions: The interface mode, which is set via drive parameter P2038, has not been set to SIMODRIVE 611 universal.
The alarm can be disabled by $MN_DRIVE_FUNCTION_MASK - bit15.
However, the following must be noted:
- The device-specific assignment of the bits in the control and status words may be different.
- The drive data sets can be created at will, and need not be subdivided into groups of 8 (for details see also SINAMICS Commissioning Manual). So the parameters of motors 2-4 may be incorrectly assigned.

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

Remedy: - Set P2038 = 1 or
- Set P0922 = 100...199 or
- Set bit 15 of $MN_DRIVE_FUNCTION_MASK (note the boundary conditions, see above)
and execute a Power ON in each case.

Program Continuation: Switch control OFF - ON.

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 was invalid because a changed gear stage ratio was found between encoder and processing during the parameter block change.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Please inform the authorized personnel/service department.
Re-referencing/resynchronization of the absolute encoder; attach encoder on the load side and configure correctly (e.g. MD 31040 ENC_IS_DIRECT).

Program Continuation: Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group.
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.

Reaction: The NC switches to follow-up mode.
Local alarm reaction.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: -

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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 MD 36400 CONTOUR_TOL setting was over-sensitive).

Reaction: Alarm display.

Remedy: 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.

Reaction: 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.

26072 Channel %1 axis %2 cannot be controlled by the PLC

Parameters: %1 = channel number
%2 = axis name, spindle number

Definitions: The axis cannot be made a PLC-controlled axis. For the time being, the axis cannot be controlled at any state from the PLC.

Reaction: Interface signals are set.
Alarm display.

Remedy: Use Release or Waitp to make the axis a neutral one.

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

" Siemens AG 2007 All Rights Reserved

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.

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

Remedy: Activate axial RESET and repeat procedure.

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

26075 Channel %1 axis %2 not available for the NC program, as exclusively controlled by the PLC

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

Definitions: The axis is exclusively controlled by the PLC. Therefore, the axis is not available for the NC program.

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

Remedy: Let the PLC control the axis not exclusively, but only temporarily. Change machine date $MA_BASE_FUNCTION_MASK bit 4.

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

26076 Channel %1 axis %2 not available for NC program, firmly assigned PLC axis

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

Definitions: The axis is a firmly assigned PLC axis. The axis is therefore not available for the NC program.

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

Remedy: Do not define axis as a firmly assigned PLC axis. Change of machine date $MA_BASE_FUNCTION_MASK bit5.

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

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.

Reaction: Alarm display.

Remedy: Preset value by means of POLFA(Axis,Type,Pos), with type = 1 (absolut) 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.

Reaction: Alarm display.

" Siemens AG 2007 All Rights Reserved

Remedy: Preset axis PLC-controlled (declare single axis).

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

26082 Channel %1 ESR for PLC-controlled axis %2 has been triggered

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

Definitions: An axial ESR has been triggered for an individual axis (PLC-controlled axis):
The display can be suppressed by machine date MD 11410: SUPPRESS_ALARM_MASK bit28 = 1.

Reaction: Alarm display.

Remedy: The individual axis is in axial stop after the ESR movement.
If an axial reset is performed for the individual axis, the alarm will be deleted
and the individual axis can be traversed again.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.
The individual axis is in axial stop after the ESR movement.
If an axial reset is performed for the individual axis, the alarm will be deleted
and the individual axis can be traversed again.

26100 Axis %1 drive %2 sign of life missing

Parameters: %1 = axis name, spindle number
%2 = drive number

Definitions: The drive control cell counter increases by one “sign of life cell” in each control cycle. The servo checks this counter for changes in the interpolation cycle. If the number of cells remains unchanged, the alarm is set.

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

Remedy: When this alarm is triggered, other error messages (system errors, e.g. stack overflow) are displayed. Evaluation of these messages provides information on the cause of the error.
If this alarm occurs repeatedly, the alarm displays and an exact as possible machine and program analysis should be recorded and reported to the Siemens AG, system support hotline for A&D MC products.

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.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Check bus configuration.
Check the interface (connector removed, option module inactive, etc.).

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
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.

Remedy: Check cycle setting, extend cycle time if required.
Restart drive, 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 SDB.

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

Remedy: Possible causes:
MD 30130 CTRLOUT_TYPE not equal to 0 as a result of an oversight; the drive should actually be simulated (= 0).
MD 30110 CTRLOUT_MODULE_NR entered incorrectly, i.e. the logical drive numbers have been mixed up, or a drive number which does not exist on the bus has been entered (check e.g. the number of slaves)
An incorrect SDB is being used or the addresses of the input and output slots of the drives were not selected during 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 SDB.

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

Remedy: Possible causes:
MD 30240 ENC_TYPE not equal to 0 as a result of an oversight; the encoder should actually be simulated (= 0).
MD 30220 ENC_MODULE_NR entered incorrectly, i.e. the logical drive numbers have been mixed up, or a drive number which does not exist on the bus has been entered (check e.g. the number of slaves)
An incorrect SDB is being used or the addresses of the input and output slots of the drives were not selected during PROFIBUS configuration.

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

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).

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

Remedy: Reboot the drive, hardware reset.

Program Continuation: Switch control OFF - ON.

26120 Channel %1 axis %2 $AA_ESR_ENABLE = 1 but axis should be set to NEUTRAL

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

Definitions: One axis with ESR configuration and $AA_ESR_ENABLE[Achse] = 1 should be set to NEUTRAL. However, neutral axes (apart from single axes) cannot execute an ESR.

Reaction: Alarm display.

Remedy: Set $AA_ESR_ENABLE[Achse] = 0 before setting axis to NEUTRAL.
Alarm can be suppressed via $MN_ALARM_SUPPRESS_MASK_2 bit 6 = 1.

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

26121 Channel %1 axis %2 is NEUTRAL and $AA_ESR_ENABLE = 1 should be set

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

Definitions: $AA_ESR_ENABLE[Achse] = 1 should not be set to neutral axes (apart from single axes). Neutral axes (apart from single axes) cannot execute an ESR.

Reaction: Alarm display.

Remedy: Do not apply $AA_ESR_ENABLE[Achse] = 1 to neutral axes (apart from single axes).
Alarm can be suppressed via $MN_ALARM_SUPPRESS_MASK_2 bit 6 = 1.

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

26122 Channel %1 axis %2, $AA_ESR_ENABLE = 1, axis replacement not executed in this state

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

Definitions: With $AA_ESR_ENABLE[Achse] = 1 axis replacement not permitted.

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

Remedy: Set $AA_ESR_ENABLE[axis] = 0 before axis replacement.

Program Continuation: Clear alarm with the RESET key. Restart part program
Set $AA_ESR_ENABLE[axis] = 0

" Siemens AG 2007 All Rights Reserved

26123 Channel %1 axis %2, $AA_ESR_ENABLE = 1 should be set, but $MA_ESR_REACTION = 0

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

Definitions: $AA_ESR_ENABLE[axis] = 1 should only be set on axes with $MA_ESR_REACTION[Achse] > 0.
The following example brings about the alarm:
N100 $MA_ESR_REACTION[AX1] = 21
N110 $AA_ESR_ENABLE[AX1] = 1
N120 NEWCONF
because $MA_ESR_REACTION[AX1] = 21 will become known to the NCK at the time of N120 NEWCONF.
Correct would be:
N100 $MA_ESR_REACTION[AX1] = 21
N110 NEWCONF
N120 $AA_ESR_ENABLE[AX1] = 1

Reaction: Alarm display.

Remedy: Before setting $AA_ESR_ENABLE[axis] = 1, $MA_ESR_REACTION[axis] > 0 must be set.
When setting $MA_ESR_REACTION[axis] in the parts program, e.g. NEWCONF must be called before $AA_ESR_ENABLE[axis].
Alarm can be suppressed via $MN_ALARM_SUPPRESS_MASK_2 bit 6 = 1.

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

26124 Channel %1 axis %2, $AC_ESR_TRIGGER triggered but axis is NEUTRAL and cannot execute ESR

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

Definitions: Channel-specific ESR ($AC_ESR_TRIGGER) triggered, but one axis with ESR configuration is NEUTRAL at the time of triggering.
Neutral axes are ignored with ESR (apart from single axes which react only to $AA_ESR_TRIGGER[Ax]).

Reaction: Alarm display.

Remedy: $AA_ESR_ENABLE[Achse] = 1 should not be set with neutral axes.
Alarm can be suppressed via $MN_ALARM_SUPPRESS_MASK_2 bit 6 = 1.

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

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 the NC or neutralized.

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

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

" Siemens AG 2007 All Rights Reserved

2.2 Cycle alarms

61000 Channel %1 block %2: No tool offset active

Parameters:

Definitions: Source (cycle):
SLOT1, SLOT2
POCKET3, POCKET4
CYCLE71
CYCLE72
CYCLE93 to CYCLE95

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

Remedy: A tool must be programmed with corrections in the program called up.

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

61001 Channel %1 block %2: Thread lead incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE84
CYCLE840
CYCLE97
CYCLE376T

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

Remedy: Check parameters for thread size and check pitch information (contradict each other)

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

61002 Channel %1 block %2: Type of machining incorrectly defined

Parameters:

Definitions: Source (cycle):
SLOT1, SLOT2
POCKET3, POCKET4
CYCLE71
CYCLE72
CYCLE93
CYCLE95
CYCLE97

Remedy: The machining type parameter VARI has been set to the wrong value and needs to be altered.

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

" Siemens AG 2007 All Rights Reserved

61003 Channel %1 Block %2: No feed programmed in cycle

Parameters:

Definitions: Source (cycle):
CYCLE71
CYCLE72
CYCLE371T to CYCLE374T
CYCLE383T to CYCLE385T
CYCLE381M, CYCLE383M, CYCLE384M, CYCLE387M

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

Remedy: The parameter for feedrate has been incorrectly set and must be altered.

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

61004 Channel %1 Block %2: Incorrect configuration of geometry axes

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

Definitions: The geometry-axes sequence is wrong. CYCLE328

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

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.

Reaction: 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

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.

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61009 Channel %1 Block %2: Active tool number = 0

Parameters:

Definitions: Source (cycle):
CYCLE71
CYCLE72

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

Remedy: No tool (T) has been programmed before the cycle call.

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

61010 Channel %1 Block %2: Finishing allowance too large

Parameters:

Definitions: Source (cycle):
CYCLE71
CYCLE72

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

Remedy: An active scaling factor is active which is not permissible for this cycle.

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

61011 Channel %1 Block %2: Scaling not permissible

Parameters:

Definitions: Source (cycle):
CYCLE72

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

Remedy: The finishing allowance at the base is larger than the overall depth, it must be reduced.

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

61012 Channel %1 Block %2: Different scaling in planes

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

Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77.

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

Remedy: --

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

" Siemens AG 2007 All Rights Reserved

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.

Reaction: 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

61015 Channel %1 Block %2: Contour is not defined

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61017 Channel %1 block %2: function %4 not present in NCK

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61018 Channel %1 block %2: function %4 not executable with NCK

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61019 Channel %1 Block %2: Parameter %4 incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: CYCLE60, CYCLE83.

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

Remedy: Check the value of the parameter.

" Siemens AG 2007 All Rights Reserved

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

61020 Channel %1 block %2: Machining not possible with active TRANSMIT/TRACYL

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61021 Channel %1 block %2: Parameter %4 value too high

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61022 Channel %1 block %2: Parameter %4 value too low

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61023 Channel %1 block %2: Parameter %4 value must be unequal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

" Siemens AG 2007 All Rights Reserved

61024 Channel %1 bock %2: Parameter %4 check value

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61025 Channel %1 block %2: Check tool carrier position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61026 Channel %1 block %2: Cycle cannot be executed with NC function %4.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

61099 Channel %1 block %2: Internal cycle error (%4)

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: .

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

Remedy:

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

" Siemens AG 2007 All Rights Reserved

61101 Channel %1 block %2: Reference plane incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE71
CYCLE22
CYCLE81 to CYCLE88
CYCLE840
CYCLE375T
SLOT1, SLOT2
POCKET3, POCKET4

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

Remedy: Either different values must be entered for the reference plane and the retraction plane if they are relative values or an absolute value must be entered for the depth.

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

61102 Channel %1 block %2: No spindle direction programmed

Parameters:

Definitions: Source (cycle):
CYCLE86
CYCLE88
CYCLE840
CYCLE370T to CYCLE374T, CYCLE376T
CYCLE383T to CYCLE385T
CYCLE381M, CYCLE383M, CYCLE384M, CYCLE387M
POCKET3, POCKET4

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

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 holes is zero

Parameters:

Definitions: Source (cycle):
HOLES1
HOLES2

Remedy: No value has been programmed for the number of holes.

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

61104 Channel %1 block %2: Contour violation of grooves

Parameters:

Definitions: Source (cycle):
SLOT1
SLOT2

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

Remedy: Errors in parameterization of milling pattern in those parameters which define the position and shape of slots/longitudinal holes on a circle

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

" Siemens AG 2007 All Rights Reserved

61105 Channel %1 block %2: Milling cutter radius too large

Parameters:

Definitions: Source (cycle):
SLOT1, SLOT2
POCKET3, POCKET4

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

Remedy: The cutter diameter in the tool offset memory is larger than the pocket or slot width.
Use a smaller cutter or change the pocket width

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

61106 Channel %1 block %2: Number of or distance between circular elements

Parameters:

Definitions: Source (cycle):
HOLES2
SLOT1, SLOT2

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

Remedy: Incorrect parameterization of NUM or INDA.
The layout of the circle elements in a full circle is not possible.

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

61107 Channel %1 block %2: First drilling depth incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE83

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

Remedy: Change value for first drilling depth (first drilling depth is in the opposite direction to the total drilling depth)

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

61108 Channel %1 Block %2: Values for parameters _RAD1 and _DP1 not permissible

Parameters:

Definitions: Source (cycle):
POCKET3
POCKET4

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

Remedy: The parameters _RAD1 and _DP for defining the path for the depth infeed have been incorrectly set.

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

" Siemens AG 2007 All Rights Reserved

61109 Channel %1 Block %2: Parameter _CDIR incorrectly defined

Parameters:

Definitions: Source (cycle):
POCKET3
POCKET4

Remedy: The cutting direction _CDIR parameter has been set to the wrong value and needs to be altered.

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

61110 Channel %1 Block %2: Finishing allowance at bottom > depth infeed

Parameters:

Definitions: Source (cycle):
POCKET3
POCKET4

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

Remedy: The finishing allowance at the base has been specified to be larger than the maximum depth infeed; either reduce the finishing allowance or increase the depth infeed.

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

61111 Channel %1 Block %2: Infeed width > Tool diameter

Parameters:

Definitions: Source (cycle):
CYCLE71
POCKET3
POCKET4

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

Remedy: The programmed infeed width is greater than the diameter of the active tool. It must be reduced.

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

61112 Channel %1 Block %2: Tool radius negative

Parameters:

Definitions: Source (cycle):
CYCLE72

Remedy: The radius of the active tool is negative. This is not permitted.

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

61113 Channel %1 Block %2: Parameter _CRAD too large for corner radius

Parameters:

Definitions: Source (cycle):
POCKET3

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

Remedy: The parameter for the corner radius _CRAD has been set too large. It must be decreased.

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

" Siemens AG 2007 All Rights Reserved

61114 Channel %1 Block %2: Machining direction G41/G42 incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE72

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

Remedy: The machining direction of the cutter radius compensation G41/G42 has been incorrectly selected.

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

61115 Channel %1 Block %2: Approach or retract mode(straight / circle / plane / space) incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE72

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

Remedy: An incorrect contour approach or retract mode was defined; check parameter _AS1 or _AS2.

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

61116 Channel %1 Block %2: Approach or retract path = 0

Parameters:

Definitions: Source (cycle):
CYCLE72

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

Remedy: The approach or retract path is set to zero, it must be increased; check parameter _LP1 and/or _LP2.

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

61117 Channel %1 Block %2: Active tool radius <= 0

Parameters:

Definitions: Source (cycle):
CYCLE71
POCKET3
POCKET4

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

Remedy: The radius of the active tool is negative or zero. This is not permitted.

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

61118 Channel %1 Block %2: Length or width = 0

Parameters:

Definitions: Source (cycle):
CYCLE71

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

" Siemens AG 2007 All Rights Reserved

Remedy: The length or width of the cutting area is not permitted; 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 programmed incorrectly

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.
Reaction: 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: Thread type inside / outside 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.
Reaction: 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 is 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.
Reaction: 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 distance incorrectly defined in plane

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The safety clearance is negative or zero. This is not allowed.
Reaction: 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

" Siemens AG 2007 All Rights Reserved

61123 Channel %1 Block %2: CYCLE72 cannot be simulated

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

Definitions: Alarm triggered by following cycle: CYCLE72.

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

Remedy:

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

61124 Channel %1 Block %2: Infeed width is not programmed

Parameters:

Definitions: Source (cycle):
CYCLE71

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

Remedy: In active simulation without a 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.

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

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.

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

Remedy: Program lower spindle speed/raise reference plane.

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

61127 Channel %1 block %2: Wrong definition of tapping axis transformation ratio (machine data)

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

Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840.

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

Remedy: Check machine data 31050 and 31060 in the appropriate gear stage of the drilling axis.

" Siemens AG 2007 All Rights Reserved

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

61128 Channel %1 block %2: Insertion angle = 0 for insertion with oscillation or helix

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

Definitions: Alarm triggered by following cycle: SLOT1.

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

Remedy: Check parameter _STA2.

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

61129 Channel %1 block %2: perpendic. approach and retraction during contour milling only allowed with G40

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

Definitions: Alarm triggered by following cycle: CYCLE72.

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

Remedy:

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

61130 Channel %1 block %2: positions of parallel axes cannot be compensated. No workpiece reference agreed.

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61131 Channel %1 block %2: parameter _GEO incorrect, _GEO=%4

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

" Siemens AG 2007 All Rights Reserved

61132 Channel %1 block %2: parallel axis parameter incorrect, check values for parallel axis parameters ABS/INK

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61133 Channel %1 block %2: 3rd parallel axis parameter incorrect, check axis name or GUD _SCW_N[]

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61134 Channel %1 block %2: rotary axis parameter incorrect, check values for rotary axis parameters ABS/INK

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61135 Channel %1 block %2: incorrect parameter sequence for approaching target position: %4

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

" Siemens AG 2007 All Rights Reserved

61136 Channel %1 block %2: no 3rd geometry axis agreed in GUD _SCW_N[]

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61137 Channel %1 block %2: swiveling and parallel axes cycle are mutually exclusive because of workpiece reference $P_WPFRAME

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

61138 Channel %1 block %2: parameter %4 incorrectly defined for tool monitoring in cycles

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

Definitions:

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

Remedy:

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

61139 Channel %1 block %2: error in function Tool monitoring in cycles

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

Definitions: Alarm triggered by following cycle: CYCLE69.

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

Remedy:

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

" Siemens AG 2007 All Rights Reserved

61175 Channel %1 block %2: angle of aperture _DF programmed too small

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

Definitions: The angle of aperture of the text in the engraving cycle is too small. This means that the text for engraving does not fit in the specified angle.

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

Remedy: Enter a larger angle of aperture.

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

61176 Channel %1 block %2: text length _DF programmed too small

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

Definitions: The text length in the engraving cycle is too short. This means that the text for engraving is longer than the specified text length.

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

Remedy: Enter longer text length

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

61177 Channel %1 block %2: polar text length > 360 degrees

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

Definitions: In the engraving cycle, the polar text length must not exceed 360 degrees.

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

Remedy: Enter shorter text length.

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

61178 Channel %1 block %2: code page not present

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

Definitions: The specified code page is not supported by the cycle.

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

Remedy: Use code page 1252.

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

61179 Channel %1 block %2: character does not exist, no.: %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Character number

Definitions: The character entered in the text for engraving cannot be milled.

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

" Siemens AG 2007 All Rights Reserved

Remedy: Enter another character.
Program Continuation: Clear alarm with the RESET key. Restart part program

61185 Channel %1 block %2: no or wrong (min > max) angle areas of rotary axes agreed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The rotary axis angle range is invalild. Alarm triggered by following cycles: CYCLE800.
Reaction: 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

61187 Channel %1 block %2: block search mode not allowed -> select block search with contour calculation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The alarm is triggerd by the following cycles: CYCLE800.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select block search with contour calculation.
Program Continuation: Clear alarm with the RESET key. Restart part program

61191 Channel %1 block %2: 5 axis transformation not set up

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The alarm is triggerd by the following cycles: CYCLE832.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Program Continuation: Clear alarm with the RESET key. Restart part program

61192 Channel %1 block %2: second 5 axis transformation not set up

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: The alarm is triggerd by the following cycles: CYCLE832.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61193 Channel %1 block %2: compressor option not set up

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

Definitions: The alarm is triggerd by the following cycles: CYCLE832.

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

Remedy: --

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

61194 Channel %1 block %2: spline interpolation option not set up

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

Definitions: The alarm is triggerd by the following cycles: CYCLE832.

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

Remedy: --

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

61197 Channel %1 block %2: no swiveling in JOG --> active WO G%4 and basic frames contain rotations

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

Definitions: Alarm triggered by following cycles: CYCLE800.

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

Remedy: --

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

61198 Channel %1 block %2: no swiveling in JOG --> several active basic frames(G500) contain rotations

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

Definitions: Alarm triggered by following cycles: CYCLE800.

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

Remedy: --

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

61199 Channel %1 block %2: approach of tool and swivel data record change (TOOLCARRIER) not allowed

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

Definitions: Alarm triggered by following cycles: CYCLE800.

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

" Siemens AG 2007 All Rights Reserved

Remedy: --

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

61201 Channel %1 block %2: Wrong sequence in 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: E_CP_CE, E_CP_DR, E_MANAGE, F_CP_CE, F_CP_DR, F_MANAGE.

Reaction: 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: E_MANAGE, F_MANAGE.

Reaction: 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 position cycle

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

Definitions: No positioning cycle was programmed in the machining block.
Alarm triggered by following cycles: E_MANAGE, F_MANAGE.

Reaction: 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

61204 Channel %1 block %2: Technology cycle unknown

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.

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61205 Channel %1 block %2: Position cycle unknown

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.

Reaction: 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 exisit.
Alarm triggered by following cycles: E_MANAGE, E_PS_CIR, E_PS_MRX, E_PS_POL, E_PS_SEQ, E_PS_XYA, F_MANAGE, F_PS_CIR, F_PS_MRX, F_PS_SEQ

Reaction: Interpreter stop
NC Start disable in this channel.
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: An incremental indication was made, but the absolute reference is unknown.
Alarm triggered by following cycles: E_MI_CON, E_MI_PL, E_PI_CIR, E_PI_REC, E_PO_CIR, E_PO_REC, E_PS_CIR, E_PS_HIN, E_PS_MRX, E_PS_POL, E_PS_SEQ, E_PS_XYA, E_SL_CIR, E_SL_LON, F_PS_CIR, F_PS_MRX, F_PS_SEQ

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

Remedy: Program an absolute position prior to using incremental indications.

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

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_O1, E_DR_PEC, E_DR_SIN, E_MI_TXT, F_DR, F_DR_PEC, F_DR_SIN, F_DRILL, F_DRILLC, F_DRILLD, F_DRM_DR, F_DRM_PE, F_DRM_SI, F_GROOV, F_MI_TXT, F_MT_LEN, F_PARTOF, F_ROU_Z, F_ROUGH, F_SP_EF, F_TAP, F_TR_CON, F_UCUT_T

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61214 Channel %1 block %2: No lead programmed

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

Definitions: No lead/helical lead has been entered.
Alarm triggered by following cycles: E_CR_HEL, E_PO_CIR, E_PO_REC, F_PO_CIR, F_PO_REC.

Reaction: 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

61216 Channel %1 Block %2: Feed/tooth only possible with cutting 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.

Reaction: 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 programmed for tool radius 0

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_DR_SIN, E_DR_TAP, E_TFS, F_DR_SIN, F_DR_TAP, F_DRILLC, F_DRM_TA, F_TAP, F_TFS

Reaction: 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 tools equals 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.

Reaction: 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

61219 Channel %1 Block %2: Tool radius too large

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

" Siemens AG 2007 All Rights Reserved

Definitions: The tool radius is too large for machining.

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

Remedy: Select a suitable tool.

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

61220 Channel %1 Block %2: Tool radius too small

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

Definitions: The tool radius is too small for machining.
Alarm triggered by following cycles: CYCLE78.

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

Remedy: Select a suitable tool.

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

61221 Channel %1 Block %2: No tool active

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

Definitions: No tool active.

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

Remedy: Select a suitable tool.

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

61222 Channel %1 Block %2: Plane infeed greater than 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_MI_PL, E_PO_CIR, E_PO_REC, F_PO_CIR, F_PO_REC.

Reaction: 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

61223 Channel %1 Block %2: Approach path too small

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.

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61224 Channel %1 Block %2: Retract path too small

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.

Reaction: 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 record 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.
Alarm triggered by following cycles: E_TCARR, F_TCARR.

Reaction: 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: Inclinable 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.
Alarm triggered by following functions: E_TCARR, F_TCARR.

Reaction: 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 "Rotary axes" to "Automatic" or "Manual".

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

61231 Channel %1 block %2: ShopMill program %4 not executable, as not tested by ShopMill

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Program name

Definitions: Before a ShopMill program can be executed, it has to be tested by ShopMill.
Alarm triggered by following cycle: E_HEAD.

Reaction: 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: Impossible to load magazine tool

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

" Siemens AG 2007 All Rights Reserved

Definitions: Only manual tools may be loaded into a swivel head in which only manual tools can be loaded.
The alarm is triggered by the following cycles: E_TD, E_TFS, F_TFS

Reaction: 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 angle wrongly defined

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

Definitions: The thread angles were specified too large or too small.
Alarm triggered by following cycles: E_TR_CON, F_TR_CON

Reaction: 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

61234 Channel %1 block %2: ShopMill subroutine %4 cannot be executed, as not tested by ShopMill

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Subroutine name

Definitions: Before a ShopMill subroutine can be used, it has to be tested by ShopMill.
Alarm triggered by following cycle: E_HEAD.

Reaction: 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 not tested by ShopTurn.

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Program name

Definitions: Before a ShopTurn program can be executed, it has to be tested by ShopTurn.
Alarm triggered by following cycle: F_HEAD

Reaction: 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 not tested by ShopTurn.

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Subroutine name

" Siemens AG 2007 All Rights Reserved

Definitions: Before a ShopTurn subroutine can be used, it has to be tested by ShopTurn.
Alarm triggered by following cycle: F_HEAD.

Reaction: 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. Withdraw tool manually!

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

Definitions: The tool is in the retraction area and it is unknown in which direction it can be travelled out of it.
Alarm triggered by following cycle: F_SP_RP

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

Remedy: Manually retract the tool from the retraction area defined in the program header and restart the program.

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

61238 Channel %1 Block %2: Machining direction unknown!

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

Definitions: The direction of the next machining is unknown.
Alarm triggered by following cycle: F_SP_RP.

Reaction: 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 within retraction area!

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

Definitions: The tool change point has to be far enough outside the retraction area so that when the revolver is swiveled, no tool extends into the retraction area.
The alarm is triggered by the following cycle: F_SP_RP

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61240 Channel %1 Block %2: Wrong feed type

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

Definitions: The feed type is not possible for this machining.
Alarm triggered by following cycles: F_DRM_DR, F_DRM_PE, F_DRM_RE, F_DRM_SI, F_GROOV, F_MIM_TR, F_ROUGH, F_SP_EF, F_UCUT_T

Reaction: 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: Retraction plane not defined for this machining direction

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

Definitions: No retraction plane has been defined for the selected machining direction.
Alarm triggered by following cycles: F_SP_RP, F_SP_RPT.

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

Remedy: Define the missing retraction plane.

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

61242 Channel %1 block %2: Wrong machine direction

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

Definitions: The machining direction has been specified incorrectly.
Alarm triggered by following cycles: F_DR, F_DR_PEC, F_DR_REA, F_DR_SIN, F_DR_TAP, F_DRILL, F_DRILLC, F_DRILLD, F_DRM_DR, F_DRM_PE, F_DRM_RE, F_DRM_SI, F_DRM_TA, F_MI_CON, F_MI_EDG, F_MI_TR, F_MI_TXT, F_MIM_TR, F_PI_CIR, F_PI_REC, F_PO_CIR, F_PO_REC, F_SL_CIR, F_SL_LON, F_TAP.

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

Remedy: Check the programmed machining direction.

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

61243 Channel %1 block %2: Correct tool change point, tool tip in

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

Definitions: The tool change point must be situated so far outside the retraction area that no tool protrudes into the retraction area on turret swivelling.
Alarm triggered by following cycle: F_SP_RP

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61244 Channel %1 block %2: Pitch change causing

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

Definitions: The specified lead change causes a reversal of the thread direction.
Alarm triggered by following cycle: CYCLE99

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

Remedy: Check lead change and thread geometry.

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

61245 Channel %1 block %2: Machining plane does not match modal

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

Definitions: Machining plane does not match modal one.

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

Remedy: Check the machining plane.

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

61246 Channel %1 block %2: Safety distance too small

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

Definitions: The safety clearance is too small for machining.
Alarm triggered by following cycle: CYCLE79.

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

Remedy: Increase safety clearance.

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

61247 Channel %1 block %2: Blank radius too small

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

Definitions: The blank radius is too small for machining.
Alarm triggered by following cycle: CYCLE79.

Reaction: 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

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

Definitions: The infeed is too small for machining.
Alarm triggered by following cycle: CYCLE79.

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

" Siemens AG 2007 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

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

Definitions: The number of edges is too small.
Alarm triggered by following cycle: CYCLE79.

Reaction: 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: Width across flats/edge length too small

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

Definitions: The width across flats/edge length is too small.
Alarm triggered by following cycle: CYCLE79.

Reaction: 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: Width across flats/edge length too large

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

Definitions: The width across flats/edge length is too large.
Alarm triggered by following cycle: CYCLE79.

Reaction: 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

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

Definitions: Chamfer/radius is too large.
Alarm triggered by following cycle: CYCLE79.

Reaction: 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

" Siemens AG 2007 All Rights Reserved

61253 Channel %1 Block %2: No finishing allowance programmed

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

Definitions: No finishing allowance has been entered.
Alarm triggered by following cycles: E_PO_CIR, E_PO_REC, E_SL_CIR, E_SL_LON, F_PO_CIR, F_PO_REC, F_SL_CIR, F_SL_LON.

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

Remedy: Programm a finishing allowance.

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

61254 Channel %1 Block %2: Error while traveling to fixed stop

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

Definitions: Error on travelling to fixed stop.
Alarm triggered by following cycle: F_SUB_SP.

Reaction: 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: Clear alarm with the RESET key. Restart part program

61255 Channel %1 block %2: Error during cut-off: Tool broken?

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

Definitions: Cut-off could not be completed. A tool breakage might have occurred.
Alarm triggered by following cycles: F_PARTOF, F_SUB_SP.

Reaction: 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

61256 Channel %1 block %2: Mirroring not allowed at program start. Deselect work offset!

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

Definitions: Mirroring impermissible at program start.
Alarm triggered by following cycle: F_HEAD.

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

Remedy: Deselect work offset.

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

" Siemens AG 2007 All Rights Reserved

61257 Channel %1 block %2: incomplete installation of counterspindle

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

Definitions: Start-up of the counterspindle is incomplete.
Alarm triggered by following cycle: F_SUB_SP.

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

Remedy: Check display machine data 9803, 9851, 9852, 9853 and 9854 .

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

61258 Channel %1 block %2: set parameters for counterspindle chuck in the spindle image

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

Definitions: The parameters for the counterspindle chuck have not been set in the spindle view.
Alarm triggered by following cycle: F_SUB_SP.

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

Remedy: Specify parameters "ZL1", "ZL2" and "ZL3" in mask "Tools work offset" > "Spindles".

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

61259 Channel %1 block %2: program contains new machining steps from ShopMill %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = ShopMill version

Definitions: The program has been created with a ShopMill version that is higher than the existing one.

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

Remedy: Delete the machining step and reprogram machining if required.

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

61260 Channel %1 block %2: program contains new machining steps from ShopTurn %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = ShopTurn version

Definitions: The program has been created with a ShopMill version that is higher than the existing one.

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

Remedy: Delete the machining step and reprogram machining if required.

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

" Siemens AG 2007 All Rights Reserved

61261 Channel %1 block %2: center offset too large

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

Definitions: The center offset on center drilling is larger than permissible.
Alarm triggered by following cycles: F_DRILL, F_DRILLD.

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

Remedy: Enter smaller center offset (see display machine data 9862).

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

61262 Channel %1 block %2: lead not possible with selected tool

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

Definitions: The lead of the tap does not match the programmed lead.
Alarm triggered by following cycles: F_DR_TAP, F_DRM_TA, F_TAP.

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

Remedy: Use a tap with the programmed lead.

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

61263 Channel %1 Block %2: Chained ShopMill program blocks not permissible in subprogram on pos. pattern

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

Definitions: If a subroutine is called from a position pattern, the subroutine itself must not include a position pattern.
The alarm is triggered by the following cycle: E_MANAGE

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

Remedy: Reprogram machining.

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

61264 Channel %1 Block %2: Chained ShopTurn program blocks not permissible in subprogram on pos. pattern

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

Definitions: If a subroutine is called from a position pattern, the subroutine itself must not include a position pattern.
Alarm triggered by following cycle: F_MANAGE.

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

Remedy: Reprogram machining.

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

" Siemens AG 2007 All Rights Reserved

61265 Channel %1 block %2: Too many restrictions, use rectangular pocket

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

Definitions: In face milling a maximum of only 3 sides can be delimited.
Alarm triggered by following cycle: CYCLE61

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

Remedy: Use pocket cycle.

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

61266 Channel %1 Block %2: Illegal machining direction

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

Definitions: In face milling, the delimitations and the direction of machining do not match.
Alarm triggered by following cycle: CYCLE61

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

Remedy: Select another direction of machining.

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

61267 Channel %1 Block %2: Plane infeed too large, residual corners remain

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

Definitions: In face milling, the plane infeed must not exceed 85%.
Alarm triggered by following cycle: CYCLE61

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

Remedy: Select a smaller plane infeed, as otherwise residual corners will be left over.

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

61268 Channel %1 block %2: Illegal machining direction, residual corners are left over.

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

Definitions: In face milling, the machining direction does not match the selected delimitations.
Alarm triggered by following cycle: CYCLE61.

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

Remedy: The machining direction must be selected to match the delimitations.

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

" Siemens AG 2007 All Rights Reserved

61269 Channel %1 block %2: External tool diameter too small

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

Definitions: Incorrect tool definition.
Alarm triggered by following cycle: CYCLE61.

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

Remedy: Check angle and diameter of the tool used.

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

61270 Channel %1 block %2: Chamfer width too small

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

Definitions: Chamfer width selected too small.
Alarm triggered by the following cycles: E_SP_CHA, F_SP_CHA.

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

Remedy: Increase the chamfer width.

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

61271 Channel %1 block %2: Chamfer width > tool radius

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

Definitions: Chamfer width larger than tool radius.
Alarm triggered by following cycles: E_SP_CHA, F_SP_CHA.

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

Remedy: Use a larger tool.

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

61272 Channel %1 block %2: Insertion depth too small

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

Definitions: Insertion depth on chamfering too small.
Alarm triggered by following cycles: E_SP_CHA, F_SP_CHA.

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

Remedy: Increase the insertion depth.

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

" Siemens AG 2007 All Rights Reserved

61273 Channel %1 block %2: Insertion depth too large

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

Definitions: Insertion depth on chamfering too large.
Alarm triggered by following cycles: E_SP_CHA, F_SP_CHA.

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

Remedy: Decrease the insertion depth.

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

61274 Channel %1 block %2: Invalid tool angle

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

Definitions: Invalid tool angle.
Alarm triggered by following cycles: E_SP_CHA, F_SP_CHA.

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

Remedy: Check tool angle.

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

61275 Channel %1 block %2: Target point violates software limit switch!

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

Definitions: Due to a swivel action, the end point is outside the software limit switches.
Alarm triggered by following cycle: E_SP_RP.

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

Remedy: Select another retraction plane or approach a suitable interpolation point.

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

61276 Channel %1 block %2: External tool diameter required for restrictions

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

Definitions: Outer tool diameter required in case of delimitations.
Alarm triggered by following cycle: CYCLE61.

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

Remedy: Specify the outer tool diameter.

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

61277 Channel %1 block %2: Tool diameter larger than restriction

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

Definitions: Tool diameter larger than delimitation.
Alarm triggered by following cycle: CYCLE61.

Reaction: Interpreter stop

" Siemens AG 2007 All Rights Reserved

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

Remedy: Use a smaller tool.

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

61278 Channel %1 block %2: If tool angle is larger than 90°, both tool diameters must be equal

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

Definitions: For tool angles larger than 90°, the two tool diameters must be identical.
Alarm triggered by following cycle: CYCLE61.

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

Remedy: Correct the tool angle or the tool diameters.

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

61279 Channel %1 block %2: If tool angle equals 90°, both tool diameters must be equal

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

Definitions: For tool angles equal to 90°, the two tool diameters must be identical.
Alarm triggered by following cycle: CYCLE61.

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

Remedy: Correct the tool angle or the tool diameters.

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

61280 Channel %1 Block %2: Mirroring in WO %4 missing

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

Definitions: If the program starts with a counterspindle movement, a work offset with mirroring will have to be selected.
Alarm triggered by following cycle: F_SUB_SP

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

Remedy: Select the mirroring for the work offset used.

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

61281 Channel %1 block %2: starting point of machining outside retraction planes

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

Definitions: The starting point of machining is outside the retraction planes.
Alarm triggered by following cycle: F_SP_RP.

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

Remedy: Adjust the retraction planes.

" Siemens AG 2007 All Rights Reserved

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

61282 Channel %1 block %2: end point of machining outside retraction planes

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

Definitions: The end point of machining is outside the retraction planes.
Alarm triggered by following cycle: F_SP_RP.

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

Remedy: Adjust the retraction planes.

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

61283 Channel %1 block %2: direct approach not possible, as tool change required

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

Definitions: After block search a position is to be reached by direct approach, but a tool change is required before.
Alarm triggered by following cycle: F_TFS.

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

Remedy: First execute a manual tool change, then restart the block search.

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

61284 Channel %1 block %2: starting point cannot be approached without collision. Pre-position tool manually

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

Definitions: The starting point cannot be approached without collisions.
Alarm triggered by following cycles: F_DRILL, F_DRILLC, F_DRILLD, F_DRM_DR, F_DRM_PE, F_DRM_RE, F_DRM_SI, F_DRM_TA, F_GROOV, F_MIM_TR, F_PARTOF, F_SP_EF, F_TAP, F_TR_CON, F_UCUT_T.

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

Remedy: Preposition the tool manually.

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

61285 Channel %1 block %2: parking position is below return plane XRA.

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

Definitions: The parking position is below retraction plane XRA.
Alarm triggered by following cycle: F_SP_RP.

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

Remedy: Move the parking position above retraction plane XRA.

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

" Siemens AG 2007 All Rights Reserved

61286 Channel %1 block %2: machining not possible, check tool angle.

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

Definitions: Machining not possible with the specified tool.
Alarm triggered by following cycles: F_UCUT_T.

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

Remedy: Use a suitable tool.

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

61287 Channel %1 block %2: no master spindle active.

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

Definitions: No master spindle active.
Alarm triggered by following cycle: F_TFS.

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

Remedy: Activate the master spindle (machine data 20090).

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

61300 Channel %1 Block %2: Probe defective

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

Definitions:

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

Remedy:

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

61304 Channel %1 Block %2: Allowance

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

Definitions:

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

Remedy:

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

61305 Channel %1 Block %2: Dimension too small

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

Definitions:

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

Remedy:

" Siemens AG 2007 All Rights Reserved

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

61321 Channel %1 Block %2: Check WO memory number

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

Definitions: WO with the number specified in _KNUM not existing. Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998.

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

Remedy: Check parameter _KNUM.

Program Continuation: Clear alarm with the RESET key. Restart part program

61337 Channel %1 Block %2: Check measuring input

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61346 Channel %1 Block %2: Distance starting point/measuring point _SETV[0] and _SETV[1] <=0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE961.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Parameters _SETV[0] or _SETV[1] are empty or smaller than 0.

Program Continuation: Clear alarm with the RESET key. Restart part program

61348 Channel %1 Block %2: Angle rel. to reference edge equals 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61357 Channel %1 Block %2: No resources free

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
Not enough NC memory space available.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Delete the files.

Program Continuation: Clear alarm with the RESET key. Restart part program

61369 Channel %1 block %2: Position of corner not clearly definable, check parameter (_SETV[0...7])

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE961.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Define P1 and P2 or P3 and P4 in a way that the intersection of the straights determined by these points is outside the sections formed by P1 and P2 or P3 and P4.

Program Continuation: Clear alarm with the RESET key. Restart part program

61501 Channel %1 block %2: Simulation is active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: all grinding cycles

Remedy: Reset simulation

Program Continuation: Clear alarm with the RESET key. Restart part program

61502 Channel %1 block %2: No tool offset active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: all grinding cycles

Remedy: A tool number must be programmed

Program Continuation: Clear alarm with the RESET key. Restart part program

61503 Channel %1 block %2: tool nose radius compensation left or right

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE410, CYCLE411, CYCLE412, CYCLE413, CYCLE414, CYCLE415, CYCLE416, CYCLE420

Remedy: A tool offset value has to be programmed

Program Continuation: Clear alarm with the RESET key. Restart part program

61504 Channel %1 block %2: _KNG incorrect for setup

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61505 Channel %1 block %2: retraction path < 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE420

Remedy: Increase retraction path

Program Continuation: Clear alarm with the RESET key. Restart part program

61506 Channel %1 block %2: infeed path < 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE420

Remedy: Increase infeed path

Program Continuation: Clear alarm with the RESET key. Restart part program

61507 Channel %1 block %2: safety clearance < 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61508 Channel %1 block %2: Incorrect default setting for shoulder position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61509 Channel %1 block %2: Incorrect default setting for dresser position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61510 Channel %1 block %2: Test run feed is active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE410, CYCLE411, CYCLE413, CYCLE415, CYCLE420

Remedy: Switch off test run feed

Program Continuation: Clear alarm with the RESET key. Restart part program

61511 Channel %1 block %2: Incorrect shoulder position or tool edge D1/D2

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61512 Channel %1 block %2: Incorrect longitudinal position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61513 Channel %1 block %2: Dresser left and inclined grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61514 Channel %1 block %2: Grinding wheel type missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: setup function

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61515 Channel %1 block %2: Retraction path <= dressing amount

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE416

Remedy: Change retraction path

Program Continuation: Clear alarm with the RESET key. Restart part program

61517 Channel %1 block %2: Angle of inclined grinding wheel missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE416

Remedy: Enter angle under $TC_TPG8

Program Continuation: Clear alarm with the RESET key. Restart part program

61518 Channel %1 block %2: Shoulder height of grinding wheel must be > grinding wheel radius

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE432

Remedy: Change shoulder height or grinding wheel radius

Program Continuation: Clear alarm with the RESET key. Restart part program

61519 Channel %1 block %2: Incorrect type of machining

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE410, CYCLE411, CYCLE412, CYCLE413, CYCLE415

Remedy: Assign a value between 1 and 3 to parameter B_ART

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61520 Channel %1 block %2: Additional offsets not set

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE413, CYCLE420, CYCLE433

Remedy: Set MD18094 MM_NUM_CC_TDA_PARAM=10

Program Continuation: Clear alarm with the RESET key. Restart part program

61521 Channel %1 block %2: Current grinding wheel too wide

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE411, CYCLE415

Remedy: Reduce width of grinding wheel

Program Continuation: Clear alarm with the RESET key. Restart part program

61522 Channel %1 block %2: Overlap >= current grinding wheel width

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE411

Remedy: Reduce overlap

Program Continuation: Clear alarm with the RESET key. Restart part program

61523 Channel %1 block %2: Zero signal of calipers missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE410, CYCLE411, CYCLE413

Remedy: Check calipers signal

Program Continuation: Clear alarm with the RESET key. Restart part program

61524 Channel %1 block %2: Incorrect oblique angle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE413

Remedy: Oblique plunge angles must be >-90° and <90°

Program Continuation: Clear alarm with the RESET key. Restart part program

61525 Channel %1 block %2: Incorrect grinding wheel type

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE413

Remedy: Change grinding wheel type $TC_TPC1

Program Continuation: Clear alarm with the RESET key. Restart part program

61526 Channel %1 block %2: Workpiece radius = 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE414

Remedy: Enter workpiece radius > 0

Program Continuation: Clear alarm with the RESET key. Restart part program

61527 Channel %1 block %2: Grinding wheel radius >= workpiece radius

Parameters: %1 = Channel number
%2 = Block number, label channel number

" Siemens AG 2007 All Rights Reserved

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE414
Remedy: Change grinding wheel radius or workpiece radius
Program Continuation: Clear alarm with the RESET key. Restart part program

61529 Channel %1 block %2: Dimensional notation INCH programmed

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE410, CYCLE411, CYCLE412, CYCLE413, CYCLE414, CYCLE415, CYCLE420
Remedy: Basic system MD $MN_SCALING_SYSTEM_IS_METRIC does not correspond to programmed G command (G group 13).
Program Continuation: Clear alarm with the RESET key. Restart part program

61530 Channel %1 block %2: Default longitudinal position incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE420
Remedy: Check longitudinal position parameter
Program Continuation: Clear alarm with the RESET key. Restart part program

61531 Channel %1 block %2: Longitudinal position not registered in Z

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE420
Remedy: Increase infeed path parameter
Program Continuation: Clear alarm with the RESET key. Restart part program

61532 Channel %1 block %2: Value for _LAGE is incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycle: CYCLE414
Remedy: Correct parameter content for _LAGE
Program Continuation: Clear alarm with the RESET key. Restart part program

61533 Channel %1 block %2: No length L1 entered under D...

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE416, CYCLE420
Remedy: Enter length L1 in the tool offset D of the grinding wheel
Program Continuation: Clear alarm with the RESET key. Restart part program

61540 Channel %1 block %2: Incorrect D number / dresser D field active

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE401, CYCLE402, CYCLE403, CYCLE443
Remedy: A tool D number must be programmed that is < _GC_DNUM
Program Continuation: Clear alarm with the RESET key. Restart part program

61541 Channel %1 block %2: Incorrect grinding wheel type entered

Parameters: %1 = Channel number
%2 = Block number, label channel number

" Siemens AG 2007 All Rights Reserved

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE432, CYCLE434, CYCLE435, CYCLE436, CYCLE438, CYCLE439, CYCLE444, CYCLE447

Remedy: Select a valid grinding wheel type in tool management

Program Continuation: Clear alarm with the RESET key. Restart part program

61542 Channel %1 block %2: Incorrect grinding wheel reference point selected when selecting the dresser coordinate system

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE435, CYCLE441, CYCLE447

Remedy: A tool D number must be programmed that is < _GC_DNUM

Program Continuation: Clear alarm with the RESET key. Restart part program

61543 Channel %1 block %2: Incorrect dresser selected when selecting the dresser coordinate system

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE402, CYCLE435, CYCLE442, CYCLE447

Remedy: A dresser number >0 and <4 must be selected

Program Continuation: Clear alarm with the RESET key. Restart part program

61544 Channel %1 block %2: Grinding wheel diameter worn down

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE438

Remedy: New grinding wheel required, or check limit values in the grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61545 Channel %1 block %2: Width of grinding wheel worn down

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE438

Remedy: New grinding wheel required, or check limit values in the grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61546 Channel %1 block %2: Dresser %4, wear limit length 1 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE438

Remedy: New dresser required, or check limit values of dresser

Program Continuation: Clear alarm with the RESET key. Restart part program

61547 Channel %1 block %2: Dresser %4, wear limit length 2 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE438

Remedy: New dresser required, or check limit values of dresser

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61548 Channel %1 block %2: Dresser %4, wear limit length 3 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE438

Remedy: New dresser required, or check limit values of dresser

Program Continuation: Clear alarm with the RESET key. Restart part program

61549 Channel %1 block %2: Incorrect dresser type selected

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycles: CYCLE402, CYCLE421, CYCLE422, CYCLE423, CYCLE424

Remedy: Check dresser type on input

Program Continuation: Clear alarm with the RESET key. Restart part program

61555 Channel %1 block %2: Diameter of grinding wheel ==0, GWPS cannot be calculated

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE446

Remedy: Check diameter

Program Continuation: Clear alarm with the RESET key. Restart part program

61556 Channel %1 block %2: Impossible chamfer and radius of left edge of wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE432

Remedy: Check values in grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61557 Channel %1 block %2: Impossible chamfer and radius of right edge of wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE432

Remedy: Check values in grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61558 Channel %1 block %2: Chamfer / radius + shoulder height are less than the retraction height of the left edge of the grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE432

Remedy: Check values in grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61559 Channel %1 block %2: Chamfer / radius + shoulder height are less than the retraction height of the right edge of the grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following grinding cycle: CYCLE432

" Siemens AG 2007 All Rights Reserved

Remedy: Check values in grinding wheel data

Program Continuation: Clear alarm with the RESET key. Restart part program

61601 Channel %1 block %2: Finished part diameter too small

Parameters:

Definitions: Source (cycle):
CYCLE94

Remedy: A finished part diameter of <3mm has been programmed. Increase value.

Program Continuation: Clear alarm with the RESET key. Restart part program

61602 Channel %1 block %2: Tool width incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE93

Remedy: The tool width (recessing tool) is larger than programmed recess width.

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61603 Channel %1 block %2: Recess type incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE93
CYCLE374T

Remedy: (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.

Program Continuation: Clear alarm with the RESET key. Restart part program

61604 Channel %1 block %2: Active tool violates programmed contour

Parameters:

Definitions: Source (cycle):
CYCLE95

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Contour violation in relief cut elements caused by the tool clearance angle of the tool used, i.e. use another tool or check sub-contour program.

Program Continuation: Clear alarm with the RESET key. Restart part program

61605 Channel %1 block %2: Contour incorrectly programmed

Parameters:

Definitions: Source (cycle):
CYCLE95

Remedy: Impermissible relief cut element detected

Program Continuation: Clear alarm with the RESET key. Restart part program

61606 Channel %1 block %2: Error during contour preparation

Parameters:

Definitions: Source (cycle):
CYCLE95

Remedy: Check sub-contour program
This alarm is always triggered in conjunction with a 10930 ... 10934, 15800 or 15810 NCK alarm.

Program Continuation: Clear alarm with the RESET key. Restart part program

61607 Channel %1 block %2: Starting point incorrectly programmed

Parameters:

Definitions: Source (cycle):
CYCLE95
CYCLE376T

Remedy: The starting point reached before the cycle call does not lie outside the rectangle described by the contour subroutine.

Program Continuation: Clear alarm with the RESET key. Restart part program

61608 Channel %1 block %2: Incorrect tool point direction programmed

Parameters:

Definitions: Source (cycle):
CYCLE94

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

" Siemens AG 2007 All Rights Reserved

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: Shape incorrectly defined

Parameters:

Definitions: Source (cycle):
CYCLE94

Remedy: Check parameters for undercut form.

Program Continuation: Clear alarm with the RESET key. Restart part program

61610 Channel %1 Block %2: No infeed depth programmed

Parameters:

Definitions: Source (cycle):
CYCLE374T

Remedy: Change infeed depth.

Program Continuation: Clear alarm with the RESET key. Restart part program

61611 Channel %1 Block %2: No point of intersection found

Parameters:

Definitions: Source (cycle):
CYCLE95

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: No intersection could be calculated with the contour. Check contour programming or change infeed depth.

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.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check value in parameter _VARI.

Program Continuation: Clear alarm with the RESET key. Restart part program

61710 Channel %1 block %2: Stock removal program not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61711 Channel %1 block %2: Name of stock removal program missing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61712 Channel %1 block %2: Tool parameter for machining direction not defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61766 Channel %1 block %2: Error in blank program

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61798 Channel %1 block %2: Acknowledgment error ACTIVATE

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61799 Channel %1 block %2: Acknowledgment error READYPROG

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with the RESET key. Restart part program

61800 Channel %1 block %2: Ext. CNC system missing

Parameters:

Definitions: Source (cycle):
CYCLE328,
CYCLE370T to CYCLE374T, CYCLE376T,
CYCLE383T to CYCLE385T,
CYCLE381M, CYCLE383M, CYCLE384M,
CYCLE387M

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Machine data for external language MD 18800 MM_LANGUAGE or option bit 1919800 ON_EXTERN_LANGUAGE is not set.

Program Continuation: Clear alarm with the RESET key. Restart part program

61801 Channel %1 block %2: Wrong G code selected

Parameters:

Definitions: Source (cycle):
CYCLE370T to CYCLE374T, CYCLE376T,
CYCLE383T to CYCLE385T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: An impermissible numerical value has been programmed in the CYCLE ... <value> program call, or an incorrect value was specified in the cyclic setting data for the G code system. Correct the values.

Program Continuation: Clear alarm with the RESET key. Restart part program

61802 Channel %1 block %2: Wrong axis type

Parameters:

Definitions: Source (cycle):
CYCLE328

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The programmed axis is assigned to a spindle.

Program Continuation: Clear alarm with the RESET key. Restart part program

61803 Channel %1 block %2: Programmed axis not available

Parameters:

Definitions: Source (cycle):
CYCLE328

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The programmed axis is not in the system. Check MD 20050 to MD20080

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61804 Channel %1 block %2: Progr. position exceeds reference point

Parameters:

Definitions: Source (cycle):
CYCLE328

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The programmed intermediate position or actual position is behind the reference point.

Program Continuation: Clear alarm with the RESET key. Restart part program

61805 Channel %1 block %2: Value programmed absolute and incremental

Parameters:

Definitions: Source (cycle):
CYCLE328
CYCLE371T to CYCLE374T, CYCLE376T,
CYCLE383T, CYCLE384T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The programmed intermediate position is both absolutely and incrementally programmed.

Program Continuation: Clear alarm with the RESET key. Restart part program

61806 Channel %1 block %2: Wrong axis assignment

Parameters:

Definitions: Source (cycle):
CYCLE328

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The axis-assignment sequence is wrong.

Program Continuation: Clear alarm with the RESET key. Restart part program

61807 Channel %1 block %2: Wrong spindle direction programmed (active)

Parameters:

Definitions: Source (cycle):
CYCLE384M

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The programmed spindle direction contradicts the spindle direction planned for the cycle.

Program Continuation: Clear alarm with the RESET key. Restart part program

61808 Channel %1 block %2: Final drilling depth or single drilling depth missing

Parameters:

Definitions: Source (cycle):
CYCLE381M, CYCLE383M, CYCLE384M,
CYCLE387M,
CYCLE383T to CYCLE385T

" Siemens AG 2007 All Rights Reserved

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Total depth “Z” or individual drilling depth “Q” missing in G8x block (first call)

Program Continuation: Clear alarm with the RESET key. Restart part program

61809 Channel %1 Block %2: Drill position not permissible

Parameters:

Definitions: Source (cycle):
ISO encasing cycles

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61810 Channel %1 Block %2: ISO G code not possible

Parameters:

Definitions: Source (cycle):
ISO encasing cycles

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Program Continuation: Clear alarm with the RESET key. Restart part program

61811 Channel %1 Block %2: ISO axis name illegal

Parameters:

Definitions: Source (cycle):
CYCLE370T to CYCLE374T,
CYCLE376T,
CYCLE383T to CYCLE385T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: An impermissible numerical value has been defined in the call block.

Program Continuation: Clear alarm with the RESET key. Restart part program

61812 Channel %1 Block %2: Value(s) in external cycle call wrongly defined

Parameters:

Definitions: Source (cycle):
CYCLE370T to CYCLE374T,
CYCLE376T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: An impermissible ISO axis name has been programmed in the call block.

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61813 Channel %1 Block %2: GUD value wrongly defined

Definitions: Source (cycle):
CYCLE376T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: An impermissible numerical value has been entered in the cycle setting data.

Program Continuation: Clear alarm with the RESET key. Restart part program

61814 Channel %1 block %2: Polar coordinates not possible with cycle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61815 Channel %1 block %2: G40 not active

Parameters:

Definitions: Source (cycle):
CYCLE374T,
CYCLE376T

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: G40 was inactive before the cycle call.

Program Continuation: Clear alarm with the RESET key. Restart part program

61816 Channel %1 Block %2: Axes not on reference point

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61817 Channel %1 Block %2: Axis coordinates within protection zone

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

" Siemens AG 2007 All Rights Reserved

61818 Channel %1 Block %2: Axis range limits are equal

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61912 Channel %1 block %2: No path to generate

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Program Continuation: Clear alarm with the RESET key. Restart part program

62000 Channel %1 block %2: Insert new tool

Parameters: %1 = channel number
%2 = block number, label

Definitions:

Remedy: -

Program Continuation: Clear alarm with the Delete key or NC START.

62100 Channel %1 block %2: No drilling cycle active

Parameters:

Definitions: Source (cycle):
HOLES1
HOLES2

Remedy: No modal drilling cycle has been called before the drilling pattern cycle call.

Program Continuation: Clear alarm with the Delete key or NC START.

62102 Channel %1 Block %2: pocket not completely solidly machined during finishing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Alarm display.

Remedy:

Program Continuation: Clear alarm with the Delete key or NC START.

62103 Channel %1 Block %2: No finishing allowance programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No finishing allowance is programmed, although it is necessary for this machining.

Reaction: Alarm display.

Remedy: Programm a finishing allowance.

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with the Delete key or NC START.

62104 Channel %1 Block %2: Drilling cycle incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Alarm display.

Remedy:

Program Continuation: Clear alarm with the Delete key or NC START.

62106 Channel %1 block %2: incorrect value for monitoring status in tool monitoring

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Alarm display.

Remedy:

Program Continuation: Clear alarm with the Delete key or NC START.

62107 Channel %1 block %2: parameter %4 incorrectly defined for tool monitoring in cycles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Alarm display.

Remedy:

Program Continuation: Clear alarm with the Delete key or NC START.

62108 Channel %1 block %2: error in function Tool monitoring in cycles

Parameters: %1 = Channel number
%2 = Block number, label

Definitions:

Reaction: Alarm display.

Remedy:

Program Continuation: Clear alarm with the Delete key or NC START.

62182 Channel %1 block %2 : load inclinable head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No swivel head is active. Alarm triggered by following cycles: E_TCARR, F_TCARR.

Reaction: 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 : unload inclinable head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE800.

Reaction: Alarm display.

Remedy: --

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with the Delete key or NC START.

62184 Channel %1 block %2 : replace inclinable head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE800.

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62300 Channel %1 Block %2: Check number of empirical value memory

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: --

Reaction: Alarm display.

Remedy: Check setpoint value
Increase parameter _TSA

Program Continuation: Clear alarm with the Delete key or NC START.

62304 Channel %1 Block %2: Allowance

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994.

Reaction: Alarm display.

Remedy: The difference between actual and setpoint value is larger than upper tolerance limit (parameter _TUL).

Program Continuation: Clear alarm with the Delete key or NC START.

62308 Channel %1 Block %2: Variable column width not possible

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE105.
Unable to generate variable column widths, as no header available.
A fixed column width of 12 characters is used.

Reaction: Alarm display.

Remedy: Complete the header in _PROTVAL[0].

Program Continuation: Clear alarm with the Delete key or NC START.

62309 Channel %1 Block %2: Insufficient column width

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE105.
The value to be logged is larger than the column width.

Reaction: Alarm display.

Remedy: Adjust _PROTFORM[5] or change the header at variable column width.

Program Continuation: Clear alarm with the Delete key or NC START.

62311 Channel %1 block %2: The maximum number of characters per line _PROTFORM[1] is adjusted.

Parameters: %1 = Channel number
%2 = Block number, label channel number

" Siemens AG 2007 All Rights Reserved

Definitions: The alarm is triggered by the following cycles: CYCLE105
Max. number of characters per line _PROTFORM[1] has been adjusted.
Reaction: Alarm display.
Remedy: --
Program Continuation: Clear alarm with the Delete key or NC START.

62312 Channel %1 block %2: probe is not perpendicular to plane!

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions:
Reaction: Alarm display.
Remedy: --
Program Continuation: Clear alarm with the Delete key or NC START.

62500 Channel %1 block %2: GWPS has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycle: CYCLE446
Remedy: Check the limit value for GWPS and program a lower value in the NC program if necessary
Program Continuation: Clear alarm with the Delete key or NC START.

62501 Channel %1 block %2: Speed has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycle: CYCLE446
Remedy: Check speed and program a lower value in the NC program if necessary
Program Continuation: Clear alarm with the Delete key or NC START.

62502 Channel %1 block %2: Dresser %4, GWPS has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycle: CYCLE421
Remedy: Check limit value for GWPS and program a lower value in the NC program if necessary
Program Continuation: Clear alarm with the Delete key or NC START.

62503 Channel %1 block %2: Dresser %4, speed has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycle: CYCLE421
Remedy: Check speed and program a lower value in the NC program if necessary
Program Continuation: Clear alarm with the Delete key or NC START.

62900 Channel %1 block %2: Incorrect source file

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions:
Reaction: Alarm display.
Remedy: --
Program Continuation: Clear alarm with the Delete key or NC START.

" Siemens AG 2007 All Rights Reserved

62901 Channel %1 block %2: Source file not available

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62902 Channel %1 block %2: Not yet implemented

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62903 Channel %1 block %2: Incorrect contour

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62904 Channel %1 block %2: Inconsistent tree

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62905 Channel %1 block %2: Inconsistent archive

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62906 Channel %1 block %2: Error while reading from input file

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62907 Channel %1 block %2: Error while writing to NC file

Parameters: %1 = Channel number
%2 = Block number, label channel number

" Siemens AG 2007 All Rights Reserved

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62908 Channel %1 block %2: Selfcutting contour

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62909 Channel %1 block %2: Internal error: selfcont_part

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62910 Channel %1 block %2: Error while calculating the contour orientation

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62911 Channel %1 block %2: Error on overwriting target

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62912 Channel %1 block %2: Plane cannot be specified here

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62913 Channel %1 block %2: Inch/metric indication not allowed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with the Delete key or NC START.

62914 Channel %1 block %2: Double contour pocket call

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62915 Channel %1 block %2: Contour pocket call is missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62916 Channel %1 block %2: Contour not finished

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62917 Channel %1 block %2: Contour end without specified start

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62918 Channel %1 block %2: Rapid traverse within contour definition

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62919 Channel %1 block %2: Nominal radius parameter is missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

" Siemens AG 2007 All Rights Reserved

Program Continuation: Clear alarm with the Delete key or NC START.

62920 Channel %1 block %2: Pocket surface not specified

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62921 Channel %1 block %2: Pocket depth not specified

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62922 Channel %1 block %2: Output program not specified

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62923 Channel %1 block %2: Starting point not specified

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62924 Channel %1 block %2: Too many elements in the contour

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62925 Channel %1 block %2: Radius specified together with center point

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

" Siemens AG 2007 All Rights Reserved

62926 Channel %1 block %2: Wrong radius specified

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62927 Channel %1 block %2: Error in fillet

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62928 Channel %1 block %2: Error in chamfer

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62929 Channel %1 block %2: Overlapping pockets

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62930 Channel %1 block %2: Contour not closed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62931 Channel %1 block %2: Residual material file error

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62932 Channel %1 block %2: error on reading RIF file

Parameters: %1 = Channel number
%2 = Block number, label channel number

" Siemens AG 2007 All Rights Reserved

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62933 Channel %1 block %2: DEMO mode

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Reaction: Alarm display.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

" Siemens AG 2007 All Rights Reserved

2.3 Profibus alarms

380001 Profibus-DP: startup error, reason %1 parameter %2 %3 %4.

Parameters: %1 = cause of error
%2 = parameter 1
%3 = parameter 2
%4 = parameter 3

Definitions: An error occurred during startup of the DP master.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Please proceed as follows:
Check the control project (particularly SDB); check MD 11240; if a user-specific SDB is in use, load it again.
If the error still exists, save the file and restart the control system using the default values with which the system was supplied.
If the system starts up without an error, the user data should be loaded one step at a time.
If the error still exists after startup with the default values, reboot from the PC card or update the software.
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.

380003 Profibus-DP: operating error, reason %1, parameter %2 %3 %4.

Parameters: %1 = cause of error
%2 = parameter 1
%3 = parameter 2
%4 = parameter 3

Definitions: An operating error occurred on the PROFIBUS DP in cyclic mode.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

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 (CN setting at end of cable, otherwise OFF setting required)
Check slave for error causes 02, 03, 04:
Check SDB for error causes 02, 03, 04, 05:
Follow the procedure described for troubleshooting alarm 380001
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 %3 access conflict, type %1, counter %2

Parameters: %1 = conflict type
%2 = serial number within the conflict sequence
%3 = number of the bus affected

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 Type 2. The NCK has not finished writing data when cyclic transfer begins again. Counter %2 contains a serial number starting at the number 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 during the next conflict.

Reaction: Alarm display.

Remedy: Recheck timing conditions, correctly set SYSCLOCK_CYCLE_TIME and

" Siemens AG 2007 All Rights Reserved

POSCTRL_CYCLE_DELA especially:
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: bus %3 SDB %4 error %1 source %2

Parameters: %1 = cause of error
%2 = SDB source

Definitions: Error in SDB for PROFIBUS DP configuration.
Error cause:
01=SDB not present in SDB source
02=SDB in SDB source too big.
03=SDB cannot be activated in SDB source.
SDB source:
00=Default SDB1 (selected by MD 11240 = 0 if no user SDB is loaded in the control system)
01=Standard SDB1 (selected by MD 11240 = 1)
02=Standard SDB2(selected by MD 11240 = 2)
...
100 = SDB located in supported memories (SRAM)
101 = User SDB located in file system
102 = SDB reloaded in SRAM during power-up

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check MD 11240
If SDB source = 100: Reload user SDB in passive file system /_N_IBN_DIR/_N_SDB_BIN.
If SDB source = 101, check support accumulators.
If SDB source = 102: Follow the procedure described for troubleshooting alarm 380001.
If alarm 380021 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 SDB-Type-2000 was loaded

Definitions: No user-specific SDB exists.
The default SDB 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 SDB stored in the supported RAM is lost.

Reaction: Alarm display.

Remedy: Produce user-specific SDB and load in the control or select and activate standard SDB using MD 11240 PROFIBUS_SDB_NUMBER.
Restart the NC.
If the error occurs the next time the NC is switched on, the SDB 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 bus %1 has been changed

Parameters: %1 = number of the bus affected

Definitions: The PROFIBUS configuration on the DP master was changed during operation, e.g. by downloading a new hardware configuration. Since it is possible that the cycle data has changed, operation cannot be continued and a warm start is required.

Reaction: Channel not ready.
NC Start disable in this channel.

" Siemens AG 2007 All Rights Reserved

Interface signals are set.
Alarm display.

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: bus %3, configuration error %1, parameter %2

Parameters: %1 = cause of error
%2 = parameter

Definitions: The PROFIBUS DP was not generated in SDB in accordance with the configuration specifications of the NC in use.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check that SDB
contains a diagnostic slot for every slave and
only contains slave entries that are relevant to the application.
It is possible to include a superset of slaves in SDB 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 SDB to a minimum.
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

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

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

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 = Largest axis index of control
MD 12970, 12971 PLC address range for digital inputs
MD 12978, 12979PLC address area for analog outputs
If no inconsistencies can be found in the parameters, compare the machine data with the configuration in SDB. 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 SDB.

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

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

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 = Largest axis index of control
MD 12974, 12975 PLC address range for digital outputs
MD 12982, 12983PLC address area for analog outputs

" Siemens AG 2007 All Rights Reserved

If no inconsistencies can be found in the parameters, compare the machine data with the configuration in SDB. 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 SDB.

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: SDB 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.

Reaction: Alarm display.

Remedy: Add MD or
change SDB or
disconnect the slave from PROFIBUS DP or
-acknowledge alarm

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 area requested
%2 = size of area in bytes

Definitions: An incorrect logical base address was specified for a digital input.
Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check your hardware configuration for modules that are not connected or are defective and replace them.
If the error cannot be eliminated, 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 area requested
%2 = size of 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.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check your hardware configuration for modules that are not connected or are defective and replace them.
If the error cannot be eliminated, 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 area requested
%2 = size of area in bytes

" Siemens AG 2007 All Rights Reserved

Definitions: An incorrect logical base address was set for a digital or analog output, the area resides in the access range of the PLC (process output copy, base addresses < 128).

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check your hardware configuration for modules that are not connected or are defective and replace them.
If the error cannot be eliminated, 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 bus %2 slave %1

Parameters: %1 = slave address

Definitions: Failure of a PROFIBUS slot used by the NCK for digital or analog I/O.

Reaction: 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 necessary.

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.

Reaction: Alarm display.

Remedy: For fault codes/fault values, refer to drive documentation.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380501 PROFIBUS DP: fault on bus,slave,DO-Id %1, code %2, value %3, time %4

Parameters: %1 = bus number
%2 = slave address
%3 = fault value of drive (P945)
%4 = fault time of drive (P948)

Definitions: Contents of fault memory of assigned slave.

Reaction: Alarm display.

Remedy: For fault codes/fault values, refer to drive documentation.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380502 PROFIBUS DP: bus %1, slave %2 configuration changed

Parameters: %1 = bus number
%2 = slave address

Definitions: The PB bus configuration has changed.
Causes:
- Initial start-up
- New PB slave detected on bus

Reaction: Interface signals are set.
Alarm display.

Remedy: Another warm start is needed to allow operating the bus with the new configuration.

Program Continuation: Switch control OFF - ON.

380503 PROFIBUS DP: bus %1 configuration changed

Parameters: %1 = Bus number

Definitions: A new SDB2000 with a modified configuration was provided.

" Siemens AG 2007 All Rights Reserved

	The new settings will be activated only at the next ramp-up of PROFIBUS.
Reaction:	Interface signals are set. Alarm display.
Remedy:	In order to operate the bus with the new configuration, an additional restart will be required.
Program Continuation:	Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

2.4 PLC alarms

400000 PLC STOP [Type]

Definitions: PLC not in cyclic mode. Travel with the machine is not possible.
[Type]: 1 Ready(User program has not been started)
2 Break (User program has been interrupted)
3 Error (Other PLC alarm with PLC Stop active)

Reaction: Alarm display.

Remedy: Rectify other PLC alarm;
Switch on menu in PLC stop position
or test user program.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

400002 System error [Type]

Definitions: [Type]: Type number
Internal error states are displayed with this alarm. An error number is also specified to provide further details about the cause and location of the error.

Reaction: PLC Stop

Remedy: Report this error to Siemens along with the type number.
Check cause of error in the software part named at the line number given.

Program Continuation: Switch control OFF - ON.

400004 Code error: [String] Network [No.]

Definitions: [String] : internal error code, module type
[No.] : network number
The user program contains an operation which is not supported by the control unit.

Reaction: PLC Stop

Remedy: Modify and reload user program.

Program Continuation: Switch control OFF - ON.

400005 Switch On menu in PLC stop position

Definitions: User program is not being processed

Reaction: Alarm display.

Remedy: Switch controller off/on

Program Continuation: Switch control OFF - ON.

400006 Loss of remanent PLC data

Definitions: The following causes are possible:
Control handling (e.g. standard PLC deletion, power up with default values)
Control handling of power up with backed up data without backing up data in advance
Support time exceeded

Reaction: Alarm display.

Remedy: Update the data required.

Program Continuation: Clear alarm with the Delete key or NC START.

400007 Operand error: [String] network [No.]

Definitions: [String] : Module type
[No.] : network number

Reaction: PLC Stop

Remedy: The variable displayed must be checked in the user program for violation of the address range, impermissible data type and alignment errors.

Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

400008 Programming tool - version is not compatible [Version]

Definitions: This version is not compatible with the product stage of the control system.
Reaction: PLC Stop
Remedy: Translate the user program using a suitable programming tool version and load in the control.
Program Continuation: Switch control OFF - ON.

400009 Computing time overrun at PLC level: [String] network [No.]

Definitions: [String] : module type
[No.] : network number
Check user program of the corresponding network displayed.
Reaction: PLC Stop
Remedy: Change user program
Program Continuation: Switch control OFF - ON.

400010 Arithmetic error in user program: [Type] [String] network [No.]

Definitions: Check user program in the specified network.
Type 1:Division by zero using fixed point arithmetic
Type 2:Floating point (real number) arithmetic
[String]type number, module ID
[No.]network number
Reaction: PLC Stop
Remedy: Change user program.
Program Continuation: Switch control OFF - ON.

400011 Permitted number of sub-program levels exceeded: [String] network [no.]

Definitions: [String]module ID
[No.] :network number
Check user program in the specified network.
Reaction: PLC Stop
Remedy: Change user program.
Program Continuation: Switch control OFF - ON.

400013 PLC user program is incorrect

Definitions: The PLC user program in the control is incorrect or is not available.
Reaction: PLC Stop
Remedy: Reload PLC user program.
Program Continuation: Switch control OFF - ON.

400014 PROFIBUS DP: power up interrupted, type 1 - 4

Definitions: Type 1:PROFIBUS DP power up interrupted
Type 2:Software versions of NC – PLC do not match
Type 3:Number of slots per function exceeded
Type 4:PROFIBUS DP server not ready
Reaction: PLC Stop
Remedy: Types 1 to 3:Report error to Siemens
Type 4:802D – Check and/or replace PCU hardware and/or check MD 11240
Program Continuation: Switch control OFF - ON.

" Siemens AG 2007 All Rights Reserved

400015 **PROFIBUS DP: I/O defect: log adr. [x] bus adr./slot: [y/z]**

Definitions:	The PLC-AWP is using peripheral addresses which are not available. [x]Logical I/O address [y]Bus address (slave) [z]Slot number Error causes: PROFIBUS peripheral has no voltage Slave bus address set incorrectly PROFIBUS connection faulty Active MD 11240 (SDB PROFIBUS configuration) is set incorrectly
Reaction:	PLC Stop
Remedy:	Rectify the error using the error cause
Program Continuation:	Switch control OFF - ON.

2.5 SINAMICS-Parameters

A detailed description of the SINAMICS parameters is provided in the following publication:

LH1, SINAMICS_S Parameter Manual

" Siemens AG 2007 All Rights Reserved

List of action numbers

3

The following list describes the actions stated in the alarm texts under "Action %.." according to their numbers.

No. 1, INIT

Explanation	Run Init phase (tasks are initialized after power on).
Cause	-
Remedy	-

No. 2, RESET

Explanation	Perform reset (VDI signal: Reset, mode group reset or after power on).
Cause	-
Remedy	-

No. 3, RESET_INITBLOCK

Explanation	Activate Reset Init blocks (VDI signal: Reset).
Cause	-
Remedy	-

No. 4, PROG_END

Explanation	Perform reset, end of program has been detected (NC block with M30).
Cause	-
Remedy	-

" Siemens AG 2007 All Rights Reserved

No. 5, MODESWITCHTOAPROGMODE

Explanation: Change the mode to the MDI or AUTOMATIC program operating mode
(VDI signal: BAG signal)

Cause:
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)
- 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. 6, MODESWITCHTOSAVEMODE

Explanation: Automatic change from an internal mode to the mode set externally (with TEACH_IN, an attempt is made after every stop to change from the internal mode "AUTOMATIC, MDI" to TEACH_IN).

Cause: -

Remedy: -

No. 7, MODESWITCHTOHAND-MODE

Explanation: Change the mode to a manual mode
(VDI signal (mode group): JOG, TEACH_IN, REF).

Cause
1. Nesting depth too great:
   The current processing operation can be interrupted by various events (e.g. interrupt). Depending on the event, ASUBs are activated.
   These ASUBs can be interrupted in the same manner as the user program. Unlimited nesting depth is not possible for ASUBs due to memory limitations. Example: An interrupt interrupts the current program processing. Further interrupts of higher priority interrupt processing of the previously activated asynchronous subroutines.
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.

" Siemens AG 2007 All Rights Reserved

No. 8, OVERSTOREON

Explanation Activate overstore (PI command).

Cause -

Remedy -

No. 9, OVERSTOREOFF

Explanation Activate overstore (PI command).

Cause -

Remedy -

No. 10, INTERRUPT

Explanation Perform user interrupt "ASUB" (VDI signal: Digital-analog interface, ASUB interface).

Cause
1. The channel is active due to block search or loading machine data
2. The channel has been 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, INTERRUPTFASTLIFTOFF

Explanation Perform "ASUB" user interrupt with rapid retraction (VDI signal: Digital-analog interface).

Cause See No. 10

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 12, INTERRUPTBLSYNC

Explanation Perform a user interrupt at the end of the block (VDI signal: ASUB interface, digital-analog interface).

Cause See No. 10

Remedy -

No. 13, FASTLIFTOFF

Explanation Perform a rapid retraction (VDI signal: Digital-analog interface and ASUB interface, for further actions see 10, 11, 12, 85, 86)

Cause -

Remedy -

No. 14, TM_MOVETOOL

Explanation Move tool - only with tool management (PI command).

Cause -

Remedy -

No. 15, DELDISTOGO_SYNC

Explanation Perform deletion of distance-to-go or axis synchronization (VDI signal: deletion of distance-to-go or follow-up mode) (follow-up mode: e.g. on activation of axis motion).

Cause
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

No. 16, PROGRESETREPEAT

Explanation Abort repetition of subroutine (VDI signal: Delete number of subroutine repetitions).

Cause
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

" Siemens AG 2007 All Rights Reserved

No. 17, PROGCANCELSUB

Explanation Abort subroutine processing (VDI signal: Program level abort).

Cause
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

No. 18, SINGLEBLOCKSTOP

Explanation Activate single block (VDI signal: Activate single block)

Cause -

Remedy -

No. 19, SINGLEBLOCKOFF

Explanation Deactivation of single block (VDI signal: Activate single block)

Cause -

Remedy -

No. 20, SINGLEBLOCK_IPO

Explanation Activate main run single block (OPI variable and VDI signal: Activate single block)

Cause -

Remedy -

No. 21, SINGLEBLOCK_DECODIER

Explanation Activate decoding single block (OPI variable and VDI signal: Activate single block)

Cause
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 ASUB is complete or
- Abort program

No. 22, SINGLEBLOCK_MAINBLOCK

Explanation Activate main run single block (OPI variable and VDI signal: Activate single block

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 23, SINGLEBLOCK_PATH

Explanation Activate traversing single block (OPI variable and VDI signal: Activate single block)

Cause -

Remedy -

No. 24, STARTPROG

Explanation Start program processing (VDI signal: NC Start).

Cause
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, CHANNELSTARTPROG

Explanation Start program processing (channel communication, NC block: Start).

Cause
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. (only Automatic).

Remedy
- Protect Start with WAITE
- Execute condition for clearing alarm
- Reference point approach
- Select program operating mode

No. 26, RESUMEPROG

Explanation Start continuation of program processing (VDI signal: NC Start).

Cause
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 2007 All Rights Reserved

No. 27, RESUMEJOGREFDIGIT

Explanation: Start continuation of the selected process (JOG, reference point or digitizing) (VDI signal: NC Start).

Cause:
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, STARTDIGITIZE

Explanation: Start processing in the digitizing submode (VDI signal: NC Start).

Cause:
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, STOPALL

Explanation: Stop all axes (VDI signal: Stop All or with Reset key)

Cause: -

Remedy: -

No. 30, STOPPROG

Explanation: Perform a program stop (NC block: M0)

Cause: -

Remedy: -

No. 31, STOPJOGREF

Explanation: Stop the JOG motion (VDI signal: NC Stop)

Cause: -

Remedy: -

No. 32, STOPDIGITIZE

Explanation: Stop digitizing processing (VDI signal: NC Stop)

Cause: -

Remedy: -

" Siemens AG 2007 All Rights Reserved

No. 33, STARTSIG

Explanation Start selected processing (VDI signal: NC Start).

Cause
1. Process change active (operating mode change, activate/deactivate digitizing/over-store)
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, STOPSIG

Explanation Stop the active processing (VDI signal: NC Stop)

Cause -

Remedy -

No. 35, INITIALINISTART

Explanation Start machine data processing (INI file is already in the NCK)
(PI command).

Cause -

Remedy -

No. 36, INITIALINIEXTSTART

Explanation Start machine-data processing (INI file is located externally, e.g., on MMC)
(PI command).

Cause -

Remedy -

No. 37, BAGSTOP_SLBTYPA

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.

Cause -

Remedy -

No. 38, BAGSTOPATEND_SLBTYPB

Explanation Stop because of mode group single block. VDI signal, single type B (any blocks), after stop at end of block in another channel in this mode group.

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 39, OVERSTORE_BUFFER_END_REACHED

Explanation Stop because end of overstore buffer "_N_OSTOREXX_SYF" has been reached.

Cause -

Remedy -

No. 40, PREP_STOP

Explanation Start preprocessing (NC block: Stopre)

Cause -

Remedy -

Nro. 41, PROG_STOP

Explanation Stop processing at block end (NC block: M00/M01).

Cause -

Remedy -

No. 42, STOPPROGABLOCKEND

Explanation Stop processing at block end (alarm, VDI signal: NC Stop at block limit).

Cause -

Remedy -

No. 43, STOPPROGATASUPEND

Explanation Stop at end of ASUB, if start was performed from "stopped"

Cause -

Remedy -

No. 44, PROGSELECT

Explanation Activate program (PI command)

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 45, PROGSELECTEXT

Explanation Activate the program which is still external (PI command)

Cause -

Remedy -

No. 46, CHANNEL_PROGSELECT

Explanation Program selection from another channel (channel communication, NC block: INIT).

Cause -

Remedy -

No. 47, ASUPDEFINITION

Explanation Save definition of an ASUB, which can be activated (PI command)

Cause -

Remedy -

No. 48, NEWCONF

Explanation Sets all machine data with the attribute (NEW_CONF) to active (PI command)

Cause -

Remedy -

No. 49, CLEARCANCELALARM

Explanation Clear all alarms with the clear condition CANCELCLEAR (PI command, Acknowledge Alarm key).

Cause -

Remedy -

No. 50, BLOCKSEARCHUN_CONTINUE

Explanation Continue block search (NC block, Stopre)

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 51, BLOCKSEARCHRUN_START

Explanation Start block search.(PI command)

Cause -

Remedy -

No. 52, BLOCKSEARCHRUN_RESUME

Explanation Continue block search (PI command)

Cause -

Remedy -

No. 53, DIGITIZEON

Explanation Activate digitizing (PI command)

Cause -

Remedy -

No. 54, DIGITIZEOFF

Explanation Deactivate digitizing (PI command)

Cause -

Remedy -

No. 55, FUNCTGENON

Explanation Switch on function generator (PI command)

Cause -

Remedy -

No. 56, FUNCTGENOFF

Explanation Switch off function generator (PI command)

Cause -

Remedy -

No. 57, WAITM

Explanation Wait for a program marker (channel communication, NC block: WAITM).

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 58, WAITE

Explanation Wait for end of program (channel communication, NC block: WAITE).

Cause -

Remedy -

No. 59, INIT_SYNC

Explanation Program selection from another channel, synchronously (channel communication, NC block: INIT + SYNC).

Cause -

Remedy -

No. 60, HMICMD

Explanation Wait until acknowledgement from HMI (NC block, MMC_CMD)

Cause -

Remedy -

No. 61, PROGMODESLASHON

Explanation Activate the skip-block function (VDI signal: Skip block).

Cause Nesting depth too great

Remedy
- Wait until the preceding ASUB is complete or
- Abort program

No. 62, PROGMODESLASHOFF

Explanation Deactivate the skip-block function (VDI signal: Skip block).

Cause Nesting depth too great

Remedy
- Wait until the preceding ASUB is complete or
- Abort program

" Siemens AG 2007 All Rights Reserved

No. 63, PROGMODEDRYRUNON

Explanation Activate test run (VDI signal: Rapid traverse override)

Cause
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 ASUB is complete or abort the program.
- Abort program

No. 64, PROGMODEDRYRUNOFF

Explanation Deactivate test run (VDI signal: Rapid traverse override)

Cause
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 ASUB is complete or
- Abort program

No. 65, BLOCKREADINHIBIT_ON

Explanation Activate read-in disable for main run block (VDI signal: Read-in disable).

Cause -

Remedy -

No. 66, BLOCKREADINHIBIT_OFF

Explanation Deactivate read-in disable for main run block (VDI signal: Read-in disable).

Cause -

Remedy -

No. 67, STOPATEND_ALARM

Explanation Stop at block end (alarm)

Cause -

Remedy -

No. 68, STOP_ALARM

Explanation Stop all axes (alarm)

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 69, PROGESTON

Explanation: Activate program test (VDI signal: Program test).

Cause:
1. Tool management is active
2. The NCK channel is in a state other than "ready"

Remedy:
- Backing up tool data
- Abort the program or process with the Reset key or
- wait for end of program

No. 70, PROGESTOFF

Explanation: Deactivate program test (VDI signal: Program test).

Cause: The NCK channel is in a state other than "ready"

Remedy:
- Abort the program or process with the Reset key or
- wait for end of program

No. 71, STOPATIPOBUFFER_ISEMPTY_ALARM

Explanation: Stop at the end of block preparation (alarm)

Cause: -

Remedy: -

No. 72, STOPATIPOBUF_EMPTY_ALARM_REORG

Explanation: Stop at the end of block preparation with subsequent reorganization of block processing (alarm)

Cause: Nesting depth too great

Remedy:
- Wait until the preceding ASUB is complete or
- Abort program

No. 73, CONDITIONAL_STOPATEND

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.

Cause: -

Remedy: -

" Siemens AG 2007 All Rights Reserved

No. 74, CONDITIONAL_SBL_DEC_STOPATEND

Explanation Conditional stop at block end. (Despite the start, the interpreter or the preprocessing does not manage to put a block in main run)

Cause -

Remedy -

No. 75, INTERPRETERSTOP_ALARM

Explanation Stop preprocessing (alarm)

Cause -

Remedy -

No. 76, RETREAT_MOVE_THREAD

Explanation Retraction with G33 and Stop

Cause -

Remedy -

No. 77, WAITMC

Explanation Conditional wait for program marker (NC block: WAITMC).

Cause -

Remedy -

No. 78, SETM

Explanation Set marker (NC block: SETM).

Cause -

Remedy -

No. 79, CLEARM

Explanation Clear marker (NC_block: CLEARM).

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 80, BLOCK_SELECT

Explanation Selection of an NC block (PI command)

Cause -

Remedy -

No. 81, LOCK_FOR_EDIT

Explanation Disable the NC program which is currently being processed for editing (PI command)

Cause -

Remedy -

No. 82, START_TEACHINPROG

Explanation Start a program in the TEACH IN submode (VDI signal: NC Start).

Cause See Nos. 33 and 5

Remedy -

No. 83, RESUME_TEACHINPROG

Explanation Start a program in the TEACH IN submode (VDI signal: NC Start).

Cause See Nos. 33 and 5

Remedy -

No. 84, PURE_REORG

Explanation Reorganize block processing

Cause -

Remedy -

No. 85, INTERRUPT_TOPROG_NOREPOS

Explanation Activate an "ASUB" user interrupt in a manual mode (VDI signal: ASUB interface, digital-analog interface).

Cause See No. 10

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 86, INTERRUPT_START

Explanation Activate an "ASUB" user interrupt. Is only executed if the channel is in the READY status (VDI signal: ASUB interface, digital-analog interface).

Cause See No. 10

Remedy -

No. 87, INTERRUPT_SIGNAL

Explanation Perform an "ASUB" user interrupt (VDI signal: ASUB interface, digital-analog interface; for further actions see 10, 11, 12, 85, 86)

Cause -

Remedy -

No. 88, STOPBAG

Explanation Start program processing (VDI signal: Mode group stop).

Cause -

Remedy -

No. 89, NEWCONF_PREP_STOP

Explanation Activate all machine data with the attribute (NEW_CONF) (NC_block: NEW_CONF).

Cause -

Remedy -

No. 90, BLOCKSEARCHRUN_NEWCONF

Explanation Activate all machine data with the attribute (NEW_CONF) (NC_block: NEW_CONF with block search).

Cause -

Remedy -

No. 91, CONTINUE_INTERPR

Explanation Start continuation of interpreter processing (internal preprocessing stop)

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 92, SLAVEDATA

Explanation Save interlock for data.

Cause The channel is not in the "stopped" state

Remedy -

No. 93, SET_USER_DATA

Explanation Activate user data, e.g., via MMC; newly modified tool lengths become active immediately in the running program

Cause
1. The channel is not in the "stopped" state
2. The channel has been 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, PLCVERSION

Explanation Write the user PLC version to the version file.

Cause -

Remedy -

No. 95, CONVERT_SCALING_SYSTEM

Explanation Switch measuring systems (PI command).

Cause -

Remedy -

No. 96, SYSTEM_SHUTDOWN

Explanation Shut down system (VDI signal).

Cause -

Remedy -

No. 97, SERUPRO_ON

Explanation Activate block search PI (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.

Cause -

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 98, ESR

Explanation Extended stop and retract

Cause -

Remedy -

No. 99, BLOCKSEARCHRUN_SIGNAL

Explanation Block search (general) is currently being activated (negative acknowledgement may be output for PI service).

Cause -

Remedy -

No. 100, BLOCKSEARCHRUN_INTEGR

Explanation Integrated block search, i.e., a block search is restarted on a stopped program.

Cause -

Remedy -

No. 101, EXT_ZERO_POINT

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.

Cause
1. The channel is not in AUTO or MDI
2. The channel has been stopped and the current block cannot be reorganized.

Remedy
- Select Auto or MDI
- Activate a block change until the NC block can be reorganized.

No. 102, SINGLEBLOCK_IPONOSBLOF

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.

Cause -

Remedy -

No. 103, SINGLEAX_STOPALL_MASTER

Explanation Stopping of a single axis movement (VDI signal)

Cause The axis is not controlled by the PLC. (exception old reaction with oscillation axis).

Remedy -

" Siemens AG 2007 All Rights Reserved

No. 104, SINGLEAX_STOPALARM_MASTER

Explanation	Stopping of a single axis movement by an alarm
Cause	The axis is not controlled by the PLC. (exception old reaction with oscillation axis).
Remedy	-

No. 105, SINGLEAX_RESUME_MASTER

Explanation	Continuation of a single-axis movement (VDI signal).
Cause	The axis has not been stopped previously. Not for all axis types at present.
Remedy	-

No. 106, SINGLEAX_RESET_MASTER

Explanation	Interruption of a single-axis movement (VDI signal).
Cause	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 107, SINGLEAX_DELDIS_MASTER

Explanation	Deletion of distance-to-go of a single-axis movement (VDI signal).
Cause	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 108, SINGLEAX_PLCCTRL_ON_MASTER

Explanation	Power ON: Axis is now controlled by the PLC (VDI signal).
Cause	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

No. 109, SINGLEAX_PLCCTRL_OFF_MASTER

Explanation	Deactivate: Axis is now controlled by the PLC (VDI signal).
Cause	The axis is not controlled by the PLC. Not for all axis types at present.
Remedy	-

" Siemens AG 2007 All Rights Reserved

No. 110, SINGLEAX_JOG_WHEEL

Explanation available soon

Cause

Remedy -

No. 111, SINGLEAX_JOG_PLUS_MASTER

Explanation available soon

Cause

Remedy -

No. 112, SINGLEAX_JOG_MINUS_MASTER

Explanation available soon

Cause

Remedy -

No. 113, SINGLEAX_JOG_PLUS_INC_MASTER

Explanation available soon

Cause

Remedy -

No. 114, SINGLEAX_JOG_MINUS_INC_MASTER

Explanation available soon

Cause

Remedy -

No. 115, REPOSMODECHANGE

Explanation The event is triggered by the positive PLC edge of the "Repos-Mode-Edge"signal.

Cause 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).

" Siemens AG 2007 All Rights Reserved

No. 116, TOOLCHANGECMDON

Explanation	Activate tool-management commands (Ch. VDI signal).
Cause	The NCK channel is in a state other than "ready"
Remedy	Abort the program or process with the Reset key or wait for end of program.

No. 117, TOOLCHANGECMDOFF

Explanation	Deactivate tool-management commands (Ch. VDI signal).
Cause	The NCK channel is in a state other than "ready"
Remedy	Abort the program or process with the Reset key or wait for end of program.

No. 118, SIVLIMCHANGE

Explanation	Selection of desired safety limitations (SGE) (always allowed).
Cause	-
Remedy	-

No. 119, STOPRUN

Explanation	Stop run, i.e., the NCK has automatically stopped at an OPI-defined block
Cause	1. Controller is not in automatic mode.
Remedy	-

No. 120, SINGLEAX_LIFTFAST_OFF_MASTER

Explanation	Rapid lift with a single axis
Cause	The axis is not controlled by the PLC.
Remedy	-

No. 121, SINGLEAX_STOP_LIFTOFF_MASTER

Explanation	Rapid lift with a single axis has been stopped
Cause	The axis is not controlled by the PLC and the single axis is not executing a rapid lift
Remedy	-

" Siemens AG 2007 All Rights Reserved

No. 122, TEST_SYNC_ASYNC

Explanation Only for test purposes and only in Assert systems

Cause -

Remedy -

No. 123, START_LOCK

Explanation PI _N_STRTLK Set global start disable

Cause -

Remedy -

No. 124, START_UNLOCK

Explanation PI _N_STRTUL Reset global start disable

Cause -

Remedy -

No. 125, FASTMODESWITCHTOAHANDMODE

Explanation Implicit change of operating mode to Jog at the beginning of a Jog motion in Automatic mode

See also $MN_JOG_MODE_MASK

Cause
1. A channel has exited the mode group due to an interrupt.
2. Overstore

Remedy
- Abort the program with the Reset key or wait until the interrupt is terminated
- Deselect overstore

No. 126, FASTMODESWITCHTOAPROGMODE

Explanation Implicit change of operating mode to Automatic at the end of a Jog motion in Automatic mode

See also $MN_JOG_MODE_MASK

Cause
1. A channel has exited the mode group due to an interrupt.
2. Overstore

Remedy
- Abort the program with the Reset key or wait until the interrupt is terminated
- Deselect overstore

" Siemens AG 2007 All Rights Reserved

No. 127, SIMULATIONBLOCKSEARCHRUN

Explanation: Simulation block search should be initiated, i.e., the calculation results are only displayed on the HMI and there is NO traversing even after the block search.

Cause: 1. The channel is in not in RESET

Remedy: – Press Reset

No. 128

Explanation: Refusal to execute program range

Cause: 1. The channel is in not in RESET
2. The channel is in not in Automatic

Remedy: – Press Reset
– Switch to Automatic

No. 129

Explanation: Refusal to select PI service syntax check "_N_CHKSEL"

Cause: The channel is in not in RESET

Remedy: Press Reset

No. 130

Explanation: Refusal to start PI service syntax check "_N_CHKRUN"

Cause: The channel is in not in RESET

Remedy: Press Reset

No. 131

Explanation: Refusal to start PI service syntax check "_N_CHKABO"

Cause: -

Remedy: -

No. 132

Explanation: Refusal of PI service _N_NCKMOD (BIT 1)

Cause: -

Remedy: -

" Siemens AG 2007 All Rights Reserved

No. 133

Explanation	Refusal of PI service _N_NCKMOD (BIT 1)
Cause	-
Remedy	-

" Siemens AG 2007 All Rights Reserved

" Siemens AG 2007 All Rights Reserved

Name COMPBLOCKWITHREORG

Effect Block preparation has detected an error, which can be rectified by modifying the program. Reorganization is performed after a program modification.

- Correction block with reorganization.

Name COMPENSATIONBLOCK

Effect Block preparation has detected an error, which can be rectified by modifying the program.

- Correction block

Name FOLLOWUP

Effect Follow-up of axes

- NC switches to follow-up mode

Name INTERPRETER STOP

Effect Program execution is aborted after all the prepared blocks (interpolator buffer) have been processed.

- Interpreter stop

Name LOCALREACTION

Effect

- Local alarm response

Name NOALARMREACTION

Effect

- No alarm reaction

" Siemens AG 2007 All Rights Reserved

Name 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

Name 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

Name 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

Name NONCSTART

Effect It is not possible to start a program in this channel.

- NC start inhibit in this channel

Name NOREFMARK

Effect The axes in this channel have to be rereferenced.

- Rereference axes in this channel.

Name SETVDI

Effect VDI interface signal alarm is set.

- Interface signals are set

Name SHOWALARM

Effect Alarm is displayed on MMC.

Alarm display

Name STOPBYALARM

Effect Ramp stop of all channel axes.

- NC stop for alarm

" Siemens AG 2007 All Rights Reserved

Name STOPATENDBYALARM

Effect Stop at end of block.

- NC Stop on alarm at end of block

Name 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

Name 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 view

Name ALLBAGS_NOREADY

Effect The Ready is canceled in all mode groups. The reaction thus corresponds to an NCK-REACTIONVIEW|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

Name DELAY_ALARM_REACTION

Effect If this alarm reaction is configured in the alarm handler, all alarm reactions for alarms, which occur at this point, 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 clearDelayReaction call 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

Name NO_DELAY_ALARM_REACTION

Effect The DELAY_ALARM_REACTION state is canceled.

- The alarm reaction delay is canceled.

" Siemens AG 2007 All Rights Reserved

Name 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.

4.1 Cancel criteria for alarms

Name 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

Name 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.

- The alarm is no longer displayed when the alarm cause has been removed. No other operator actions are required.

Name 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.

Name POWERONCLEAR

Effect The alarm is canceled by turning off / turning on the control system (POWER ON).

- Switch the control OFF - ON.

Name 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 the part program.

Name 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 the alarm in all channels of this mode group. Restart the part program.

" Siemens AG 2007 All Rights Reserved

Name 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 the part program.

Name NOCLEAR

Effect The clear information is only required for the internal pseudo alarm number EXBSAL_NOMOREALARMS.

" Siemens AG 2007 All Rights Reserved

" Siemens AG 2007 All Rights Reserved

Appendix A

A.1 Abbreviations

O	Output
ASCII	American Standard Code for Information Interchange: Amerikanische Code-Norm für den Informationsaustausch
AV	Job planning
BA	Operating mode
BAG	Mode groups
BB	Ready to run
BCD	Binary Coded Decimals: Decimals with each digit coded in binary
BHT	Handheld unit
UI	Operator interface
CNC	Computerized Numerical Control Computerized numerical control
CP	Communications Processor Communications processor
CPU	Central Processing Unit Central processing unit
CR	Carriage Return
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
DIO	Data Input/Output: Data transfer display
DRF	Differential resolver function : Handwheel jog
DRY	DRY run: Dry run feedrate
DSB	Decoding Single Block: Decoding single block
DSR	Data Send Ready: Signal from serial data interfaces indicating that they are ready to send
DW	Data word
E	input
EIA code	Special punched tape code, number of holes per character always odd
EPROM	Erasable Programmable Read Only Memory

" Siemens AG 2007 All Rights Reserved

I/RF	Infeed/Regenerative Feedback unit
ETC	ETC Key: Expansion of the softkey bar in the same menu
FDB	Product database
FIFO	First In First Out: Memory, which works without address specification where data are read in the same order, in which they were 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
CRC	Cutter radius compensation
FST	Feed STop: Feed stop
GUD	Global User Data : Global user data
HD	Hard Disk Hard disk
HMS	High-resolution Measuring System
MSD	Main Spindle Drive
HW	Hardware
IM	Interface Module Interface module
IM S/R	Interface Module (S=send/R=receive): Interface module for transmitting and receiving data
INC	Increment : Increment
ISO code	Special punched tape code, number of holes per character always even
K1...K4	Channel 1 to channel 4
LAD	Ladder diagram
Kv	Servo gain factor
KUE	Transmission ratio
LCD	Liquid Crystal Display: Opto-electronic display with liquid crystals
LED	Light-Emitting Diode: light-emitting-diode display
LUD	Local User Data
MB	Megabyte
MD	machine data
MC	Measuring Circuit
MDA	Manual Data Automatic: Manual input
MLFB	Machine-readable product designation
MMC	Man-Machine Communication: User interface on numerical control systems for operator control, programming and simulation
MPF	Main Program File: NC part program (main program)
MPI	Multi-Point Interface Multiple interface

" Siemens AG 2007 All Rights Reserved

MCP	Machine control panel
NC	Numerical Control: Numerical Control
NCK	Numerical Control Kernel: NC kernel with block preparation, traversing range, etc.
NCU	Numerical Control Unit: Numerical Control
NURBS	Non-Uniform Rational B-Spline
ZO	Zero Offset
OEM	Original Equipment Manufacturer
OP	Operator Panel Operator panel
OPI	Operator Panel Interface: Interface for connection to the operator panel
PC	Personal computer
PCMCIA	Personal Computer Memory Card International Association Interface standard
PG	Programming device
PLC	Programmable Logic Control: Programmable logic control
PRT	Program test
RAM	Program memory which can be read and written into
RISC	Reduced Instruction Set Computer: Type of processor with small instruction set and ability to process instructions at high speed
ROV	Rapid Override : Correction du rapide
RPA	R-Parameter Active: Memory area on the NCK for R parameter numbers
RTS	Request To Send: RTS, control signal of serial data interfaces
SBL	Single Block : Single block
SEA	Setting Data Active: Memory area for setting data on the NCK
SD	Setting Data
SKP	SKiP: Saut de bloc optionnel
SM	Signal Module
SPF	Sub Program File : Subroutine
PLC	Programmable Logic Controller (PLC)
TNRC	Tool nose radius compensation
LEC	Leadscrew error compensation
SSI	Serial Synchronous Interface: Synchronous serial interface
SW	Software
TEA	Testing Data Active: Refers to machine data
TO	Tool Offset Tool offset
TOA	Tool Offset Active: Memory area for tool offsets
TRANSMIT	Transform Milling Into Turning: Coordinate conversion on turning machine for milling operations
FDD	Feed Drive (spindle)
T	Tool

" Siemens AG 2007 All Rights Reserved

T	Tool
TO	Tool offset
ZOA	Zero Offset Active: Memory area

" Siemens AG 2007 All Rights Reserved

Index

Number

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

" Siemens AG 2006 All Rights Reserved

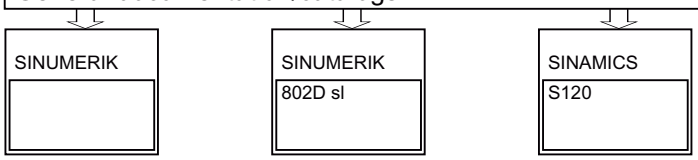
To Siemens AG A&D MC MS1 P.O. Box 3180 91050 ERLANGEN, GERMANY Fax: +49 (0) 9131 98 63 315 [documentation] mailto:docu.motioncontrol@siemens.com http://www.siemens.com/automation/service&support	**Suggestions** **Corrections** For publication/manual SINUMERIK 802D sl Diagnostics Manual User documentation
From Name: Address of your Company/Dept. Address:_____ Zip code:_____ City: _____ Phone: _____ / _____ Fax: _____ / _____ email: _____	**Diagnostics Manual** Order no.:6FC5398-2CP10-1BA0 06/2007 Edition Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections

" Siemens AG 2007 All Rights Reserved

SINUMERIK 802D sl documentation overview

General documentation/catalogs

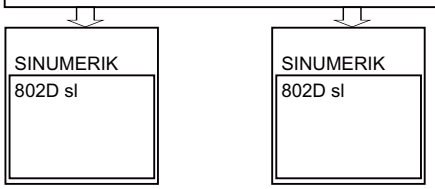


Brochure

Catalog NC 61

Catalog
D21.1 Converter
chassis units

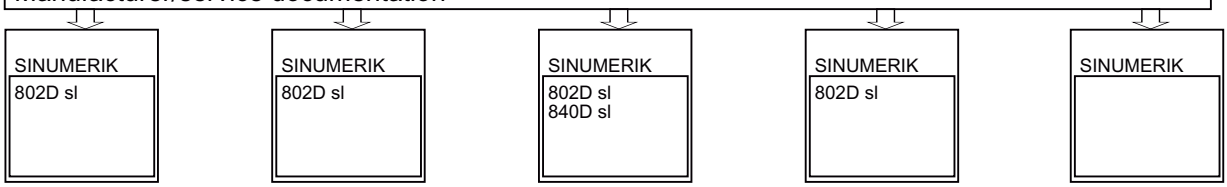
User documentation



Programming and
Operating Manual
– Turning
– Milling
– Grinding
– Nibbling

Diagnostics Manual

Manufacturer/service documentation



Operating Instructions

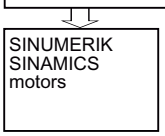
Function Manual

Function Manual
ISO Dialects

Parameter Manual

EMC Directives

Electronic documentation



DOCONCD
DOCONWEB