SIEMENS

SINUMERIK 802D sl

Diagnostics Manual

Valid for

Control system	*Software version*
SINUMERIK 802D sl	1.4

06/2009

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**
06/2009	6FC5398-2CP10-2BA0	**C**

Copyright © Siemens AG 1995 - 2009
Order No.6FC5398-2CP10-2BA0
Siemens-Aktiengesellschaft
Subject to change without prior notice

Preface

Structure of the documentation

The SINUMERIK documentation is available in three versions:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Information on the following topics is available at
http://www.siemens.com/motioncontrol/docu:

- Ordering documentation
  Here you can find an up-to-date overview of publications.
- Downloading documentation
  Links to more information for downloading files from Service & Support.
- Researching documentation online
  Information on DOConCD and direct access to the publications in DOCon-WEB.
- Compiling individual documentation on the basis of Siemens contents with the My Documentation Manager (MDM), refer to http://www.siemens.com/mdm. My Documentation Manager provides you with a range of features for generating your own machine documentation.
- Training and FAQs
  Information on the range of training courses and FAQs (frequently asked questions) are available via the page navigation.

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.

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.

Furthermore, for the sake of clarity, 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:

	Europe / Africa
Phone	+49 180 5050 222
Fax	+49 180 5050 223
€0.14/min. from German landlines, mobile phone prices may differ.	
Internet	http://www.siemens.de/automation/support-request

	America
Phone	+1 423 262 2522
Fax	+1 423 262 2200
Email	mailto:techsupport.sea@siemens.com

	Asia / Pacific
Phone	+86 1064 757575
Fax	+86 1064 747474
Email	mailto:support.asia.automation@siemens.com

Note

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

http://www.automation.siemens.com/partner

Questions about this documentation

If you have any queries (suggestions, corrections) in relation to this documentation, please send a fax or email to the following address

Fax	+49 9131 98 2176
Email	mailto:docu.motioncontrol@siemens.com

A fax form is available at the end of this document.

SINUMERIK Internet address

http://www.siemens.com/sinumerik

CompactFlash cards for the user

- The SINUMERIK CNC supports the file systems FAT16 and FAT32 for CompactFlash cards. You may need to format the memory card if you want to use a memory card from another device or if you want to ensure the compatibility of the memory card with the SINUMERIK. However, formatting the memory card will permanently delete all data on it.
- Do not remove the memory card while it is being accessed. This can result in damage to the memory card and the SINUMERIK as well as the data on the memory card.
- If you cannot use a memory card with the SINUMERIK, it is probably because the memory card is not formatted for the control system (e.g. Ext3 Linux file system), the memory card file system is faulty or it is the wrong type of memory card.
- Insert the memory card carefully and the right way round into the memory card slot (observe indicators such as arrow or similar). This way you avoid mechanical damage to the memory card or the device.
- Only use memory cards that have been approved by Siemens for use with SINUMERIK. Even though the SINUMERIK complies with the general industry standards for memory cards, it is possible that memory cards from some manufacturers will not function perfectly in this device or are not completely compatible with it (you can obtain information on compatibility from the memory card manufacturer or supplier).
- The "CompactFlash® 5000 Industrial Grade" CompactFlash card from SanDisk has been approved for SINUMERIK (Order Number 6FC5313-5AG00-0AA0).

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.

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.

Table of Contents

Overview of System Error Alarms 1

1.1 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.

Sorting

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

1.2 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"

"%" specification

The "%" specification indicates variables for an online parameter that is replaced by the corresponding value on the controller. As 802D sl is a single-channel controller system, "channel 1" is always given.

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.

1.3 Structure of alarm description

NCK alarms

Table 1-1 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

PROFIBUS alarms

Table 1-2 Number ranges of the message numbers, continued

380000 - 380502	PROFIBUS alarms

PLC alarms/messages

Table 1-3 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.

SINAMICS alarms (faults/warnings)

Tabelle 1-4 Number ranges of the message numbers

201000 - 202999	Control Unit, Regelung
203000 - 204999	Reserviert
205000 -205999	Leistungsteil
206000 - 206999	Einspeisung
207000 - 207999	Antriebe
208000 - 208999	Option Board

Tabelle 1-4 Number ranges of the message numbers

209000 - 209999	Reserviert
230000 - 230999	DRIVE-CLiQ-Komponente Leistungsteil
231000 - 231999	DRIVE-CLiQ-Komponente Geber 1
232000 - 232999	DRIVE-CLiQ-Komponente Geber 1
233000 - 233999	DRIVE-CLiQ-Komponente Geber 3
234000 - 234999	Reserviert
235000 - 235999	Terminal Module 31 (TM31)
236000 - 236999	Reserviert
250000 - 250999	Communication Board (COMM BOARD)
250400 - 265535	Reserviert

1.4 System errors

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.

Overview of alarms 2

2.1 NCK alarms

2000 PLC sign-of-life monitoring

Definitions: The PLC must give a sign of life within a defined period of time (MD10100 $MN_PLC_CYCLIC_TIMEOUT). If this does not occur, the alarm is triggered.
The sign of life is a counter reading on the internal NC/PLC interface which the PLC causes to count up with the 10 ms time alarm. The NCK also tests cyclically whether the counter reading has changed.

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: Please inform the authorized personnel/service department. Check monitoring time frame in MD10100 $MN_PLC_CYCLIC_TIMEOUT (reference value: 100ms).
Establish the cause of the error in the PLC and eliminate it (analysis of the ISTACK. If monitoring has responded with a loop in the user program rather than with a PLC Stop, there is no ISTACK entry).

Program Continuation: Switch control OFF - ON.

2001 PLC has not started up

Definitions: The PLC must give at least 1 sign of life within a period of time defined in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT (default: 1 sec.).

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: - Please inform the authorized personnel/service department. The monitoring time in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT must be checked and adapted to the first OB1 cycle.
- Determine the cause of error in the PLC (loop or stop in the user program) and eliminate it.

Program Continuation: Switch control OFF - ON.

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 (+/-15V).

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 should not occur when you remove the cable). Check the power feeder line.

Program Continuation: Switch control OFF - ON.

2900 Reboot is delayed

Definitions: This alarm indicates a delayed reboot.
This alarm only occurs when reboot was carried out by the HMI and MD10088 $MN_REBOOT_DELAY_TIME was set greater than zero.
The alarm can be suppressed with MD11410 $MN_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
Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Alarm reaction delay is cancelled.

Remedy: See MD10088 $MN_REBOOT_DELAY_TIME and MD11410 $MN_SUPPRESS_ALARM_MASK.

Program Continuation: Switch control OFF - ON.

3000 Emergency stop

Definitions: The EMERGENCY STOP request is applied to the NCK/PLC interface V2600 0000.1 (Emergency stop).

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: Please inform the authorized personnel/service department. Remove the cause of the emergency stop and acknowledge the emergency stop via the PLC/NCK interface V2600 0000.2 (emergency stop acknowledgement).

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

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

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

Definitions: The assignment of a machine axis to a channel by the MD20070 $MC_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.

Remedy: Please inform the authorized personnel/service department. The entries for the indices for the machine axes used in the channels must be contiguous in table MD20070 $MC_AXCONF_MACHAX_USED. Channel axis gaps must be enabled via MD11640$MN_ENABLE_CHAN_AX_GAP.

Program Continuation: Switch control OFF - ON.

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

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

Definitions: Only axes that have been activated in the channel by means of MD20070 $MC_AXCONF_MACHAX_USED [kx]=m may be declared as geometry axes or transformation axes by means of the MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB [gx]=k.
gx: Geometry axis index,
kx: Channel axis index,
k: Channel axis no.,
m: 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.
Correct
- MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB
- MD24... $MC_TRAFO_AXES_IN_...
- MD24... $MC_TRAFO_GEOAX_ASSIGN_TAB_...
- MD22420 $MC_FGROUP_DEFAULT_AXES
- and/or MD20070 $MC_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 MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.

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 determining a name in the NCK tables (arrays) for: machine axes, Euler angles, direction vectors, normal vectors, interpolation parameters and intermediate point coordinates, one of the following syntax rules for the identifier to be entered has been violated:
- The identifier must be an NC address letter (A, B, C, I, J, K, Q, U, V, W, X, Y, Z), possibly with a numerical extension
- The identifier must begin with any 2 capital letters but not with $ (reserved for system variables).
- The identifier must not be a keyword of the NC language (e.g. POSA).

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: MD10000 $MN_AXCONF_MACHAX_NAME_TAB
- Euler angles: MD10620 $MN_EULER_ANGLE_NAME_TAB
- Normal vectors: MD10630 $MN_NORMAL_VECTOR_NAME_TAB
- Direction vectors: MD10640 $MN_DIR_VECTOR_NAME_TAB
- Interpolation parameters: MD10650 $MN_IPO_PARAM_NAME_TAB
- Intermediate point coordinates: MD10660 $MN_INTERMEDIATE_POINT_NAME_TAB

Program Continuation: Switch control OFF - ON.

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

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

Definitions: When defining names in the channel-specific tables for geometry axes and channel axes, one of the following syntax rules for the identifier to be entered has been violated:
- The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension.
- The identifier must begin with any 2 capital letters but not with $ (reserved for system variables).
- The identifier must not be a keyword 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: MD20060 $MC_AXCONF_GEOAX_NAME_TAB
- Channel axes: MD10000 $MN_AXCONF_MACHAX_NAME_TAB

Program Continuation: Switch control OFF - ON.

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

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

Definitions: The selected identifier is invalid. Valid identifiers are:
- AX1 - AXn: Machine axis identifiers
- N1AX1 - NnAXm: Link axis identifiers (NCU + machine axis), only for 'NCU-Link' expansion level!

- C1S1 - CnSm: Container axis identifiers (container + container location), only for 'Axis container' expansion level!

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: Name of identifier
%2 = String: MD identifier

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

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: 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: Name of identifier
%3 = String: MD identifier

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

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:] Identifier missing in machine date %2[%3]

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

Definitions: An axis identifier is expected for the displayed MD in accordance with the axis configuration in the MD20070 $MC_AXCONF_MACHAX_USED and MD20050 $MC_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 the channel-specific MD20070 $MC_AXCONF_MACHAX_USED. If this concerns a geometry axis that is not to be used (this applies only for 2-axis machining, e.g. on lathes), then channel axis 0 must be entered additionally in the channel-specific MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.

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 MD20150 $MC_GCODE_RESET_VALUES or MD20100 $MC_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
%4 = There are not enough channel axes entered in the MD displayed.

Definitions: The use of the specified axis identifier in the displayed MD is not consistent the channel's axis configuration stated in the MD20070 $MC_AXCONF_MACHAX_USED and MD20050 $MC_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 MD10000 $MN_AXCONF_MACHAX_NAME_TAB, MD20080 $MC_AXCONF_CHANAX_NAME_TAB and/or MD20060 $MC_AXCONF_GEOAX_NAME_TAB.

Program Continuation: Switch control OFF - ON.

4045 [Channel %1:] Conflict between machine data %2 and machine data %3

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

Definitions: Using the specified machine data %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 lies in another type range.
NC codes/keywords can be reconfigured via machine data as long as the type range is not abandoned.
Type 1: "real" G codes: G02, G17, G33, G64, ...
Type 2: named G codes: ASPLINE, BRISK, TRANS, ...
Type 3: settable addresses: X, Y, A1, A2, I, J, K, ALF, 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 MD10712 $MN_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.: $MN_NC_USER_CODE_CONF_NAME_TAB [10] = "ROT"
$MN_NC_USER_CODE_CONF_NAME_TAB [11] = " "
clears the ROT function from the control

Program Continuation: Switch control OFF - ON.

4060 Standard machine data loaded (%1, %2, %3, %4)

Parameters: %1 = Identifier 1
%2 = Identifier 2
%3 = Identifier 3
%4 = Identifier 4

Definitions: The standard MD were loaded because
- a cold start was requested or
- the MD buffer voltage failed or
- an initialization was requested for loading the standard machine data (MD11200 $MN_INIT_MD).

Reaction: Alarm display.

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

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

4062 Backup data loaded

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

Reaction: Alarm display.

Remedy: Load specific machine data again.

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

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 with the scaling factors that can be entered (system-spefic MD array MD10230 $MN_SCALING_FACTORS_USER_DEF USER_DEF[n] (n ... index number 0 - 10), when the corresponding masking bit is set to "1".
If the masking bit is set to "0" then scaling takes place with the internal standard factors.
The following machine data influence the scaling of other MDs:
MD10220: $MN_SCALING_USER_DEF_MASK
MD10230: $MN_SCALING_FACTORS_USER_DEF

MD10240: $MN_SCALING_SYSTEM_IS_METRIC
MD10250: $MN_SCALING_VALUE_INCH
MD30300: $MA_IS_ROT_AX
If these data are modified, the NCK must be powered up again. Only then will the input of dependent data be performed correctly.

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 has been changed that affects the value of an encoder position. Please check the position values.
For absolute value encoders:
Encoder adjustment has been changed, the machine reference of the axis position may have changed, check the encoder adjustment.
Other encoders:
The reference point of the axis position has been changed, check the referencing procedure.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

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 the MD

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

Reaction: Alarm display.

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

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

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

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

Definitions: On executing a TOA file or when writing data from the part program an attempt has been made to write data with a higher protection level than the access authorization currently set in the control. The data in question have not been written and program execution is continued without hindrance. This alarm is issued on acknowledging alarm 4075. It can be cleared only with Power ON.

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

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

Program Continuation: Switch control OFF - ON.

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

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

Definitions: An attempt was made to enter a new value in the specified memory configuration machine data. It was not possible to modify the value, as this would have cleared the contents of the user memory. This was 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, for example, the LUD variables are stored and the interpolation buffer size is entered). The capacity of this memory type is defined by the current memory configuration and the value in MD18210 $MN_MM_USER_MEM_DYNAMIC.
- "S" stands for static or buffered user memory (this is where part programs, offset data, R parameters, tool data, etc. are stored). This memory type is defined by the current memory configuration and the value in MD18230 $MN_MM_USER_MEM_BUFFERED.
- "iS" stands for internal static or buffered user memory. This memory type is defined by the current memory configuration (not settable). A few NCK functions use this memory.

Reaction: Alarm display.

Remedy: If the modification was unintentional, ignore the error message and continue. The alarm has no negative effects. The remedy depends on the access right and the current memory configuration of the NCK:
- The intended change is not possible -> Try again with a smaller value. Observe the change in the number of bytes.
- Is it possible to expand the memory? This option depends on the model in use. (Not possible if parameter 4 equals "iS").
- The NCK user memory may have been set smaller than it could be. With the appropriate access authorization, the machine data (see above) can be changed.
- If parameter 4 equals "iS" and no synchronous actions are used, then MD18232 $MN_MM_ACTFILESYS_LOG_FILE_MEM[2] = 0 can be set. Otherwise the desired machine data change cannot be made.

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

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.

4110 IPO cycle changed to %1 ms

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

Definitions: The IPO cycle divisor was set to a value which was not an integral multiple of the position control cycle divisor. The divisor (MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO) was increased.
For PROFIBUS/PROFINET: MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO has been modified because of the modified DP cycle in the SDB (MD10050 $MN_SYSCLOCK_CYCLE_TIME).

Reaction: Alarm display.

Remedy: MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO has been modified.

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

4111 PLC cycle increased to %1 ms

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

Definitions: The PLC cycle divisor was set to a value which was not an integral multiple of the IPO cycle divisor. The divisor (MD10074 $PLC_IPO_TIME_RATIO) has been increased.
For PROFIBUS/PROFINET: MD10074 $PLC_IPO_TIME_RATIO has been modified because of the modified DP cycle in the SDB (MD10050 $SYSCLOCK_CYCLE_TIME).

Reaction: Alarm display.

Remedy: MD10074 $MN_PLC_IPO_TIME_RATIO has been modified.

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

4112 Servo cycle changed to %1 ms

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

Definitions: For PROFIBUS/PROFINET only:
MD10060 $POSCTRL_SYSCLOCK_TIME_RATIO has been modified because of the modified DP cycle in the SDB (MD10050 $SYSCLOCK_CYCLE_TIME).

Reaction: Alarm display.

Remedy: MD10060 $MN_POSCTRL_SYSCLOCK_TIME_RATIO RATIO has been modified.

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

4113 Sysclock cycle changed to %1 ms

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

Definitions: For PROFIBUS/PROFINET only:
MD10050 $SYSCLOCK_CYCLE_TIME has been modified because of the modified DP cycle in the SDB.

Reaction: Alarm display.

Remedy: MD10050 $MN_SYSCLOCK_CYCLE_TIME has been modified.

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

4114 Error in DP cycle of the SDB

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

Definitions: For PROFIBUS/PROFINET only:
The DP cycle in the SDB contains an error and cannot be set. The default value of MD10050$MN_SYSCLOCK_CYCLE_TIME is set.

Reaction: Alarm display.

Remedy: Correct the SDB

Program Continuation: Switch control OFF - ON.

4150 [Channel %1:] Invalid M function subprogram call configured

Parameters: %1 = Channel number

Definitions: MD10715 $MN_M_NO_FCT_CYCLE[n] or MD10718 $MN_M_NO_FCT_CYCLE_PAR contains invalid configuration data: An M function, which is occupied by the system and cannot be replaced by a subprogram call has been specified in MD10715 $MN_M_NO_FCT_CYCLE[n] for the configuration of the subprogram call via M function:
- M0 to M5,
- M17, M30,
- M19, M40 to M45,
- M function for selecting spindle/axis mode according to MD20094 $MC_SPIND_RIGID_TAPPING_M_NR (default: M70),
- M functions for nibbling/punching as configured in MD26008 $MC_NIBBLE_PUNCH_CODE if activated by MD26012 $MC_PUNCHNIB_ACTIVATION.
- Also M96 to M99 for applied external language (MD18800 $MN_MM_EXTERN_LANGUAGE).
MD10718 $MN_M_NO_FCT_CYCLE_PAR contains an invalid array index of MD10715 $MN_M_NO_FCT_CYCLE[n]. Currently, the values 0 to 9 are permissible. The affected machine data is reset to the default value -1. This deactivates the function.

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 MD10715 $MN_M_NO_FCT_CYCLE[n] that is not occupied by the system, or configure a permissible array index in MD10718 $MN_M_NO_FCT_CYCLE_PAR.

Program Continuation: Switch control OFF - ON.

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

Definitions: The "Block display with absolute values" function has been illegally parameterized:
- An illegal block length has been set with MD28400 $MC_MM_ABSBLOCK:
While ramping up, the machine data will be checked for the following value range:
0, 1, 128 to 512
- An invalid display range has been set with MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[].
While ramping up, the machine data will be checked for the following upper and lower limits:
0 <= MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[0] <= 8
0 <= MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[1] <= (MD28060 $MC_MM_IPO_BUFFER_SIZE + MD28070$MC_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.

4160 [Channel %1:] Invalid M function number configured for spindle switchover

Parameters: %1 = Channel number

Definitions: An M function was specified in MD20094 $MC_SPIND_RIGID_TAPPING_M_NR in order to configure the M function number for spindle switchover. The M function number is assigned by the system and cannot be used for the switchover (M1 to M5, M17, M30, M40 to M45).

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 MD20094 $MC_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 MD22254 $MC_AUXFU_ASSOC_M0_VALUE or MD22256 $MC_AUXFU_ASSOC_M1_VALUE, a number has been specified for the configuration of a new predefined M function which is occupied by the system, and cannot be used for an assignment. (M0 to M5, M17, M30, M40 to M45).

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 MD22254 $MC_AUXFU_ASSOC_M0_VALUE or MD22256 $MC_AUXFU_ASSOC_M1_VALUE which is not occupied by the system (M1 to M5, M17, M30, M40 to M45).

Program Continuation: Switch control OFF - ON.

4182 [Channel %1:] Invalid M auxiliary function number in %2%3, MD reset

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

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

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 = MD identifier
%4 = MD identifier

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

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 = MD identifier
%3 = If required, MD index

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

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.
See:
MD22010 $MC_AUXFU_ASSIGN_TYPE[n]
MD22020 $MC_AUXFU_ASSIGN_EXTENSION[n]
MD22030 $MC_AUXFU_ASSIGN_VALUE[n]

MD22035 $MC_AUXFU_ASSIGN_SPEC[n]

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.

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 MD20060 $MC_AXCONF_GEOAX_NAME_TAB. The channel axis number is stored with the same index in MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB. The channel axis number minus 1 provides the channel axis index under which the machine axis number is found in MD20070 $MC_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 MD30300 $MA_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 MD30310 $MA_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 MD35000 $MA_SPIND_ASSIGN_TO_MACHAX.
The channel to which this machine axis/spindle is assigned is listed in the machine axis index. (The machine axis number is given in the channel-specific MD20070 $MC_AXCONF_MACHAX_USED).

Program Continuation: Switch control OFF - ON.

4225 [Channel %1:] Axis %2 declaration as rotary axis missing

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

Definitions: The modulo functionality requires a rotary axis (positions in [deg]).

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 MD30300 $MA_IS_ROT_AX.

Program Continuation: Switch control OFF - ON.

4230 [Channel %1:] Data alteration from external not possible in current channel state

Parameters: %1 = Channel number

Definitions: It is not allowed to enter this data while the part program is being executed (e.g. setting data for working area limitation or for dry run feedrate).

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 location

Definitions: The settings for the interpolation and position control cycle were modified before the last power-up such that too little computing time is now available for the requisite cyclic task.
The alarm occurs immediately after power-up if too little runtime is available even when the axes are stationary and the NC program has not started. However, task overflow can occur only when computation-intensive NC functions are called during program execution.

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 MD10050 $MN_SYSCLOCK_CYCLE_TIME, MD10060 $MN_POSCTRL_SYSCLOCK_TIME_RATIO and/or MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO.
The test should be performed with an NC program that represents the highest possible control load. To be on the safe side, 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 MD10450 $MN_SW_CAM_ASSIGN_TAB, or several cam pairs were 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.

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. Activate required inputs/outputs via MDs:
MD10350 $MN_FASTIO_DIG_NUM_INPUTS
MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS
MD10300 $MN_FASTIO_ANA_NUM_INPUTS
MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS
Activation of fast inputs/outputs does not require the corresponding hardware configuration to be available at the control. All functions using fast inputs/outputs can also be made use of by the PLC specification/modification defined in the VDI interface, if the response time requirements are reduced accordingly.
Activated inputs/outputs increase the computation time requirement of the interpolation cycle because the PLC manipulation signals are handled cyclically. Note: Deactivate any inputs/outputs not in use.

Program Continuation: Switch control OFF - ON.

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

Parameters: %1 = String: MD identifier
%2 = String: MD identifier
%3 = No. of output

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

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.

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

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

Definitions: The axis cannot be operated as competing positioning axes,
for example because the axis is the slave axis within a closed gantry group or a gantry group to be closed.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Reset MD30450 $MA_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: MD array index

Definitions: The machine data 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: Please inform the authorized personnel/service department. Correct the MD.

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

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

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

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

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 = Block number, label
%3 = Transformation number

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

Reaction: Correction block is reorganized.802D sl
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy: Enter a valid transformation type.

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

4343 [Channel %1:] Attempt made to change the machine data of an active transformation.

Parameters: %1 = Channel number

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

Reaction: Interpreter stop
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

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

Parameters: %1 = Channel number
%2 = Name of machine data
%3 = Transformation number

Definitions: MD2.... $MC_TRAFO_GEOAX_ASSIGN_TAB_... contains an invalid entry. The following causes for the error are possible:
- The entry references a channel axis which does not exist.
- The entry is zero (no axis) but the transformation needs the relevant axis as a geometry axis.

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

Remedy: Correct the entry in MD2.... $MC_TRAFO_GEOAX_ASSIGN_TAB_ or MD2.... $MC_TRAFO_AXES_IN_.

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

4350 [Channel %1:] Axis identifier %2 machine data %3 not consistent with machine data %4

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

Definitions: MD32420 $MA_JOG_AND_POS_JERK_ENABLE (jerk limitation) and MD35240 $MA_ACCEL_TYPE_DRIVE (acceleration reduction) have been defined as the normal position 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 MD32420 $MA_JOG_AND_POS_JERK_ENABLE or MD35240 $MA_ACCEL_TYPE_DRIVE.

Program Continuation: Switch control OFF - ON.

4400 MD alteration will cause reorganization of buffered memory (Art %1), (loss of data!) - %2

Parameters: %1 = Memory type
%2 = MD identifier, if required

Definitions: A machine data has been altered that configures the buffered memory. If the NCK powers up with the altered data, this will lead to reorganization of the buffered memory and thus to the loss of all buffered user data (part programs, tool data, GUD, leadscrew error compensation, ...)
Meaning of the 1st parameter
0x00 buffered memory (internal)

0x01 buffered memory

Reaction: Alarm display.

Remedy: If the control includes user data that have 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 machine data values are overwritten by the default values at the next ramp-up. Under certain circumstances, this may cause data loss (even in the buffered 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 MD20110 $MC_RESET_MODE_MASK Bit4 and Bit5, the reset behavior of the 6th or 8th G groupe was determined. This setting is now made in MD20152 $MC_GCODE_RESET_MODE. In order to ensure compatible handling of "old" data backups, the "old" values are taken from MD20110 $MC_RESET_MODE_MASK and entered in MD20152 $MC_GCODE_RESET_MODE.

Reaction: Alarm display.

Remedy: --

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

5000 Communication job not executable %1

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

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

Reaction: Alarm display.

Remedy: - Reduce the number of communication jobs taking place at the same time or increase MD10134 $MN_MM_NUM_MMC_UNITS
- Restart communication job.
Please inform the authorized personnel/service department. No remedial measures are possible - the operation triggering the alarm message has to be repeated. Clear the alarm display with Cancel.

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

6000 Memory reorganized using standard machine data

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

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 stated to be the cause of the alarm. The MD initiating the alarm therefore has to be determined on the basis of the default values in the machine data by changing the user-specific memory structure step by step.
Usually, it is not just one single MD that has been set too large. Therefore it is advisable to reduce the memory area by a certain proportion in several MDs.

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

6010 [Channel %1:] Data block %2 not or not completely created, error code %3

Parameters: %1 = Channel number
%2 = String (block name)
%3 = Internal error code

Definitions: Data management has detected an error in power-up. The specified data block may not have been created. The error number specifies the type of error. If the error number is >100000, then there is a fatal system error. Otherwise, the user memory range was made too small. In this case the (user) error codes have the following meaning:
- Error number 1: No memory space available
- Error number 2: Maximum possible number of symbols exceeded
- Error number 3: Index 1 lies outside the valid value range
- Error number 4: Name already exists in channel
- Error number 5: Name already exists in NCK

If the alarm occurs after cycle programs, macro definitions or definitions for global user data (GUD) have been introduced, the machine data for the user memory configuration have been incorrectly configured. In all other cases, changes to machine data that are already correct lead to errors in the user memory configuration.
The following block names (2nd parameter) are known in the NCK (all system and user data blocks; in general, only problems in the user data blocks can be remedied by user intervention):
- _N_NC_OPT - System internal: option data, NCK global
- _N_NC_SEA - System internal: setting data, NCK global
- _N_NC_TEA - System internal: machine data, NCK global
- _N_NC_CEC - System internal: 'cross error compensation'
- _N_NC_PRO - System internal: protection zones, NCK global
- _N_NC_GD1 - User: 1st GUD block defined by _N_SGUD_DEF, NCK global
- _N_NC_GD2 - User: 2nd GUD block defined by _N_MGUD_DEF, NCK global
- _N_NC_GD3 - User: 3rd GUD block defined by _N_UGUD_DEF, NCK global
- _N_NC_GD4 - User: 4th GUD block defined by _N_GUD4_DEF, NCK global
- _N_NC_GD5 - User: 5th GUD block defined by _N_GUD5_DEF, NCK global
- _N_NC_GD6 - User: 6th GUD block defined by _N_GUD6_DEF, NCK global
- _N_NC_GD7 - User: 7th GUD block defined by _N_GUD7_DEF, NCK global
- _N_NC_GD8 - User: 8th GUD block defined by _N_GUD8_DEF, NCK global
- _N_NC_GD9 - User: 9th GUD block defined by _N_GUD9_DEF, NCK global
- _N_NC_MAC - User: Macro definitions
- _N_NC_FUN - System internal: predefined functions and procedures, NCK global
- _N_CHc_OPT - System internal: option data, channel-specific
- _N_CHc_SEA - System internal: setting data, channel-specific
- _N_CHc_TEA - System internal: machine data, channel-specific
- _N_CHc_PRO - System internal: protection zones, channel-specific
- _N_CHc_UFR - System internal: frames, channel-specific
- _N_CHc_RPA - System internal: arithmetic 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: toolholder data, TOA-specific
- _N_TOt_TOA - System internal: tool data, TOA-specific
- _N_TOt_TMA - System internal: magazine data, TOA-specific
- _N_NC_KIN - System internal: data to describe kinematic chains, NCK-specific
- _N_NC_NPA - System internal: data to describe 3D protection zones, NCK-specific
- _N_NC_WAL - System internal: data to describe coordinate-specific working area limitation
- _N_COMPLETE_CYD - System internal: cycle and display machine data, NCK, channel, axis specific

c = Channel number
a = Machine axis number
t = TOA unit number
There are additional internal system data blocks with 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: Correct the machine data or undo the changes made.
Please inform the authorized personnel/service department. There are two determining machine data for cycle programs:
- MD18170 $MN_MM_NUM_MAX_FUNC_NAMES = max. number of all cycle programs, error number = 2 shows that this value is too small.
- MD18180 $MN_MM_NUM_MAX_FUNC_PARAM = max. number of all parameters defined in the cycle programs, error number = 2 shows that this value is too small

(If these MDs are modified, the memory backup is retained)
The following applies to macro definitions:
MD18160 $MN_MM_NUM_USER_MACROS = max. number of all macro definitions, error number = 2 shows that this value is too small.
(If these MDs are modified, the memory backup is retained)
The following applies to GUD variables:
-
- MD18120 $MN_MM_NUM_GUD_NAMES_NCK = max. number of all NCK global GUD variables, error number = 2 shows that this value is too small.
- MD18130 $MN_MM_NUM_GUD_NAMES_CHAN = max. number of all channel-specific GUD variables in the channel, error number = 2 shows that this value is too small.
- MD18150 $MN_MM_GUD_VALUES_MEM = total value memory of all GUD variables together, error number = 1 shows that this value is too small.

Program Continuation: Switch control OFF - ON.

6020 Machine data have been 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 measures are required. Any user data that are required must be input again.

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

6030 Limit of user memory has been adapted

Definitions: Data management checks during power-up the actually available physical user memory (DRAM, DPRAM and SRAM) with the values in the system-specific machine data MD18210 $MN_MM_USER_MEM_DYNAMIC, MD18220 $MN_MM_USER_MEM_DPR und MD18230 $MN_MM_USERMEM_BUFFERED-USERMEM_BUFFERED.

Reaction: Alarm display.

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

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

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

Parameters: %1 = Free memory capacity in KB defined for the control model
%2 = Actual maximum capacity of free memory in KB
%3 = Type of memory, "D" =non-battery-backed, "S" =battery-backed

Definitions: The alarm can only occur after a 'cold start' (=NCK start-up with standard machine data). The alarm is only a notice. There is no interference with any NCK functions. It shows that the NCK has less free user memory available than specified by Siemens for this control variant. The value of the actually available free user memory can also be taken from the MD18050 $MN_INFO_FREE_MEM_DYNAMIC, MD18060 $MN_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.

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

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

6410 [TO unit %1:] Tool %2 has reached its prewarning limit with D = %4

Parameters: %1 = TO unit
%2 = Tool identifier (name)
%3 = -Not used-
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring. The actual type of tool monitoring is a tool property (see $TC_TP9). If replacement tools are not being used, the duplo number specified has no meaning. The alarm is triggered through the HMI or PLC. The channel context is not defined. The TO unit was specified for this reason (see MD28085 $MC_MM_LINK_TOA_UNIT).

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 = Channel number
%2 = Tool identifier (name)
%3 = -Not used-
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring. The actual type of tool monitoring is a tool property (see $TC_TP9).
If replacement tools are not being used, the duplo number specified has no meaning.
The alarm originates during NC program execution.

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.

6412 [TO unit %1:] Tool %2 has reached its monitoring limit with D = %4

Parameters: %1 = TO unit
%2 = Tool identifier (name)
%3 = -Not used-
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantitymonitored 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).
The alarm is triggered through the HMI or PLC.

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 = -Not used-
%4 = D number

Definitions: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.
If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring.
The actual type of tool monitoring is a tool property (see $TC_TP9).
If replacement tools are not being used, the duplo number specified has no meaning.
The alarm originates during NC program execution.

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 more cutting edges can be entered in the piece counter table.
As many cutting edges can be noted for the workpiece counter as are possible in total in the NCK. This means that if for each tool each cutting edge in each TO unit is used precisely once for a workpiece then the limit is reached.
If several workpieces are made on several toolholders/spindles simultaneously, it is possible to note MD18100 $MN_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 HMI, PLC (PI service).

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

Remedy: - Was decrementing of the piece counter forgotten? Then program SETPIECE in the part program, or add the correct command in the PLC program.
- If the part program/PLC program is correct, then more memory should be set for tool cutting edges via the MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA (can only be performed with the necessary access rights!).

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

6431 [Channel %1:] Block %2 Function not allowed. Tool management/monitoring is not active.

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

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: - Please inform the authorized personnel/service department.
- Make sure of how the NC is supposed to be configured! Is tool management or tool monitoring needed but not activated?
- Are you using a part program that is meant for a numerical control with tool management/tool monitoring? It is not possible to start this program on the numerical control without tool management/tool monitoring. Either run the part program on the appropriate NC control or edit the part program.
- Activate tool management/tool monitoring by setting the appropriate machine data. See MD18080$MN_MM_TOOL_MANAGEMENT_MASK,
MD20310$MC_TOOL_MANAGEMENT_MASK
- Check whether the required option is set accordingly.

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

6500 NC memory full

Definitions: The NCK file system is full.
The available buffered memory does not suffice. Note: At initial start-up, files of the NC file system may be affected such as drive data, HMI files, FIFO files, NC programs...

Reaction: Alarm display.

Remedy: Adjust the size of the buffered memory (MD18230 $MN_MM_USER_MEM_BUFFERED) or increase the space available in the buffered memory, e.g. by unloading part programs that are no longer being used. Or decrease the size of the ring buffer (see $MC_RESU_RING_BUFFER_SIZE).

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

6510 Too many part programs in the NC memory

Definitions: The number of files in the file system (part of the NC memory) of the NC has reached the maximum number possible. Note: During initial start-up, this can concern files from the NC file system, e.g. drive data, HMI files, FIFO files, NC programs, ...

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or unload files (e.g. part programs), or
- Increase MD18320 $MN_MM_NUM_FILES_IN_FILESYSTEM.

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

6530 Too many files in directory

Definitions: The number of files in a directory of the NC memory has reached the maximum limit.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or unload files (e.g. part programs) in the respective directory, or
- Increase MD18280 $MN_MM_NUM_FILES_PER_DIR.

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

6540 Too many directories in the NC memory

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

Reaction: Alarm display.

Remedy: - Delete or unload directory (e.g. workpiece), or
- Increase MD18310 $MN_MM_NUM_DIR_IN_FILESYSTEM.

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

6550 Too many subdirectories

Definitions: The number of subdirectories in a directory of the NCK has reached the maximum limit.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
- Delete or empty subdirectories in the respective directory, or
- Increase MD18270 $MN_MM_NUM_SUBDIR_PER_DIR.

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

6560 Data format not allowed

Definitions: An attempt was made to write impermissible data in an NCK file. This error can occur in particular when the attempt was made to load binary data in the NCK as ASCII file.
The error can also occur during preprocessing of cycles (see MD10700 $MN_PREPROCESSING_LEVEL) if the NC block is very long. In this case, subdivide the NC block.

Reaction: Alarm display.

Remedy: Specify that the file concerned is a binary file (e.g. extension: .BIN).

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

6570 NC memory full

Definitions: The NC card file system of the NCK is full. The task cannot be executed. Too many system files were created in the DRAM.

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.

6600 NC card memory is full

Definitions: The NC card file system of the NCK is full. No more data can be stored on the NC card.

Reaction: Alarm display.

Remedy: Delete the data on the NC card.

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

6610 Too many files open on NC card

Definitions: Too many files are being accessed simultaneously on the NC card.

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

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 the power is switched off while the alarm is active, the contents of the NC card are 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.

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: %1 = actManufactorCode (manufacturer code read by the card)
%2 = actDeviceCode (memory code read by the card)

Definitions: The NC card cannot be accessed because a valid write algorithm is not available for the flash memory.
Reaction: Alarm display.
Remedy: Use a compatible NC card or enter the new manufacturer code/device code in MD11700 $MN_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 the command
%4 = Programmed protection level of the command
Definitions: On assigning a protection level for a parts program command via REDEF command
- an impermissible parts program command has been programmed
- a protection level has been programmed that is logically smaller (larger in value) than the protection level currently applicable for this command.
- the relevant definition file has not been protected sufficently against write access. The write protection of the file must be at least as high as the highest protection level that has been assigned to a parts program command in this definition file.

Reaction: Alarm display.
Remedy: Modify definition files /_N_DEF_DIR/_N_MACCESS_DEF or /_N_DEF_DIR/_N_UACCESS_DEFCESS_ DEF. Please see the Siemens Programming Guide or the OEM documentation for the language commands permissible for the relevant system configurations.
Program Continuation: Clear alarm with the RESET key. Restart part program

8000 [Channel %1:] Option 'Interrupt routines' not set

Parameters: %1 = Channel number
Definitions: Fast NCK inputs are required for the input signals in order to activate the interrupt routines and rapid lift from contour. This function is not included in the basic version and must be retrofitted when needed.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Please inform the authorized personnel/service department. Do not use rapid interrupt inputs or contact the machine manufacturer with a view to retrofitting this option!
Program Continuation: Clear alarm with the RESET key. Restart part program

8010 Option 'activation of more than %1 axes' not set

Parameters: %1 = Number of axes
Definitions: More machine axes have been defined through the MD20070 $MC_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 MD20070 $MC_AXCONF_MACHAX_USED, must not exceed the maximum number of axes (dependent on configuration -> option, basic version: 4 axes).
Program Continuation: Switch control OFF - ON.

8030 [Channel %1:] Block %2 option 'interpolation of more than %3 axes' not set

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of permissible axes

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: In the part program, program maximally the number of axes required for the relevant controller configuration.

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

8040 Machine data %1 reset, corresponding option is not set

Parameters: %1 = String: MD identifier

Definitions: A machine data has been set that is locked by an option.

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. Safety functions have been selected for too many axes in the axial machine data.
The alarm can be reprogrammed in the MD11412 $MN_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: --

Program Continuation: Switch control OFF - ON.

8044 Option for IPO cycle time %1 ms not set

Parameters: %1 = Impermissible IPO cycle time

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

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 MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO)

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: One or more options were activated but no license key was set to prove the purchase of the option(s).

Reaction: Alarm display.

Remedy: Generate license key through the internet under http://www.siemens.com/automation/licence and enter it in the operating area "Start-up", function (HSK) "Licences".

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

8081 %1 option(s) is/are activated that are not licensed by the license key

Parameters: %1 = Number of non-licensed options

Definitions: One ore more options were activated, that are not licensed by the license key entered.

Reaction: NC Start disable
Alarm display

Remedy: The unlicensed options included in the option screen can be licensed using a new License Key. Licensing is described in the "Operator's Manual, Chapter: Licensing in SINUMERIK 802D sl".
Alternative: Cancel option

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 entry.

Definitions: The license key was entered wrongly at least three times. Before the next input, a new power ON is required.

Reaction: NC Start disable
Alarm display

Remedy: Perform NCK-Power-ON and enter correct License Key.

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

8100 [Channel %1:] Block %2: function not possible

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

Definitions: - Impossible due to embargo regulations:
- 1. Synchronous actions: Writing of feed, override and axial offsets ($AA_VC, $AC_VC, $AA_OVR, $AA_VC and $AA_OFF) from synchronous actions as well as Continuous Dressing can be programmed only once in a block.
- 2. Extended measurement: 'Cyclic measurement' (MEAC) and 'Measurement from synchronous action' is not possible.
- 3. Axis interpolation: The number of axes interpolating with one another must not exceed 4 (this also includes synchronous coupling of axes via synchronous actions "DO POS[X]=$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

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

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

Definitions: NC start has been activated in the MDI 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: Please inform the authorized personnel/service department. Via the channel-specific MD20700: $MC_REFP_NC_START_LOCK (NC Start without reference point) you can decide whether or not the axis has to be referenced before NC Start. The start of referencing can be enabled channel-specific or axis-specific.
Channel-specific reference point approach: The rising edge of the NC/PLC interface signal V3200 0001.0 (Activate referencing) starts an automatic sequence which starts the axes of the channel in the same sequence as specified in the axis-specific MD34110 $MA_REFP_CYCLE_NR (axis sequence channel-specific referencing). 0: The axis does not participate in channel-specific referencing, but it must be referenced for NC Start, -1: The axis does not participate in channel-specific referencing, but it need not be referenced for NC Start, 1- 8: Starting sequence for the channel-specific referencing (simultaneous start at the same no.), 1 - 31: CPU type
Axis-specific referencing: Press the direction key that corresponds to the approach direction in the axis-specific MD34010 $MA_REFP_CAM_MDIR_IS_MINUS (reference point approach in minus direction).

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

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.

10225 [Channel %1:] command denied

Parameters: %1 = Channel number

Definitions: The channel has received a command that cannot be executed.

Reaction: Alarm display.

Remedy: Press RESET.

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

10299 [Channel %1:] Auto-Repos function is not enabled

Parameters: %1 = Channel number

Definitions: The Auto-Repos function (operating mode) was selected in the channel but is not implemented.

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 remedial measures:
- Program a longer path and/or a lower traversing rate.
- Output auxiliary function in another block (program section).

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

10601 [Channel %1:] Block %2 zero velocity at block end point during thread cutting

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

Definitions: This alarm occurs only when several blocks with G33 follow in succession. The block end velocity in the specified block is zero, although a further thread cutting block follows. The reasons for this can be, for instance:
- G9
- Auxiliary function after motion
- Auxiliary function output before the motion of the following block
- Positioning axis in the block

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 MD11110 $MN_AUXFU_GROUP_SPEC [n] for selecting the output time of an auxiliary function group by changing "Auxiliary function output before/after the movement" to "Auxiliary function output during the movement".
Bit 5 = 1: Auxiliary function output before movement
Bit 6 = 1: Auxiliary function output during movement
Bit 7 = 1: Auxiliary function output after movement

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

10604 [Channel %1:] Block %2 thread pitch increase too high

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

Definitions: The thread pitch increase is causing an 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 pitch increase or path length in the NC program.

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

10605 [Channel %1:] Block %2 thread pitch decrease too high

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

Definitions: The thread pitch 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 pitch decrease or path length in the NC program.

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

10607 [Channel %1:] Block %2 thread with frame not executable

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

Definitions: The current frame is corrupting the reference between the thread length and the thread pitch.

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 a frame.

Use G63 or G331/G332.

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

10610 [Channel %1:] Axis %2 not stopped

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

Definitions: An axis/spindle has been positioned over several NC blocks using the POSA/SPOSA instruction. The programmed target position had not yet been reached ("exact stop fine" window) when the axis/spindle was reprogrammed.
Example:
N100 POSA[U]=100
:
N125 X... Y... U... ; e.g.: U axis still travels from N100!

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 keyword WAITP for axes or WAITS for spindles until the positioning axes or positioning spindles have also reached their target position.
Example for axes:
N100 POSA[U]=100
:
N125 WAITP(U)
N130 X... Y... U...
Example for spindles:
N100 SPOSA[2]=77
:
N125 WAITS(2)
N130 M6

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 traversed in the direction indicated. Exceeding the traversing range was not detected during block preparation because there has either been a motion overlay or a zero offset has been executed or a coordinate transformation is active.

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

Remedy: Depending on the reason for this alarm being triggered, the following remedial measures should be undertaken:
- Handwheel override: Cancel the motion overlay and avoid this or keep it smaller when the program is repeated.
- Transformation: Check the preset/programmed zero offsets (current frame). If the values are correct, the tool holder (fixture) must be moved in order to avoid triggering the same alarm when the program is repeated, which would again cause the program to be aborted.

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

10621 [Channel %1:] Axis %2 rests on software limit switch %3%4

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

%4 = The axis of the software limit switch is only output if different from the traversing axis.

Definitions: The specified axis is already positioned at the displayed software end delimiter.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Check machine data MD36110 $MA_POS_LIMIT_PLUS/MD36130 $MA_POS_LIMIT_PLUS2 and MD36100 $MA_POS_LIMIT_MINUS/MD36120 $MA_POS_LIMIT_MINUS2 for the software limit switches.
Shut down in JOG mode from the software limit switch.
Please inform the authorized personnel/service department.
Machine data:
Check whether the 2nd software limit switch has been selected in the.axis-specific interface signals: "V380x 1000.3 (2nd software limit switch plus) and V380x 1000.2 (2nd software limit switch minus).

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

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 other motion or do not perform overlaid motion.

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

10631 [Channel %1:] Axis %2 rests at working area limit %3%4

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = String (+ or -)
%4 = The axis of the working area limitation is only output if different from the traversing axis.

Definitions: The specified axis reaches the working area limitation in JOG mode.

Reaction: Alarm display.

Remedy: Check SD43420 $SA_WORKAREA_LIMIT_PLUS and SD43430 $SA_WORKAREA_LIMIT_MINUS for the working area limitation.

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

10650 [Channel %1:] Axis %2 incorrect gantry machine data, error code %3

Parameters: %1 = Channel number
%2 = Axis
%3 = Error no.

Definitions: An incorrect value was entered in the gantry-specific axial machine data. Further information can be derived from the error number.
- Error no. = 1 => either an incorrect gantry unit has been entered or the designation of the following axis is incorrect.
- Error no. = 2 => master axis has been specified more than once.

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 the machine data:
MD37100 $MA_GANTRY_AXIS_TYPE
0: No gantry axis
1: Master axis grouping 1
11: Slave axis grouping 1
2: Master axis grouping 2
12: Slave axis grouping 2
3: Master axis grouping 3
13: Slave axis grouping 3

Program Continuation: Switch control OFF - ON.

10651 [Channel %1:] Gantry configuration error. Error code %2

Parameters: %1 = Channel number
%2 = Reason

Definitions: The gantry configuration set in the machine data is erroneous. Gantry unit and reason for objection can be found in the transfer parameter.
The transfer parameter is made up as follows.
- %2 = error designation + gantry unit (XX).
- %2 = 10XX => no master axis declared
- %2 = 20XX => no slave axis declared
- %2 = 30XX => different contents in MD30550 $MA_AXCONF_ASSIGN_MASTER_CHAN slave axis and master axis
- %2 = 40XX => different channel or NCU assignment of the gantry axes
- %2 = 50XX => no slave axis declared in this channel
- 2% = 60XX => different channel assignment of the master axis
- %2 = 10000 => error: slave axis is geometry axis
- %2 = 11000 => error: competing positioning axis as slave axis
- %2 = 12000 => error: compile cycle axis as slave axis
- %2 = 13000 => error: gantry axis is spindle
- %2 = 14000 => error: gantry axis is Hirth geared

e.g. error code 1001 = no master axis declared, gantry unit 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.

Remedy: Please inform the authorized personnel/service department. Correct the machine data:
MD37100 $MA_GANTRY_AXIS_TYPE
0: No gantry axis
1: Master axis grouping 1
11: Slave axis grouping 1
2: Master axis grouping 2
12: Slave axis grouping 2
3: Master axis grouping 3
13: Slave axis grouping 3

Program Continuation: Switch control OFF - ON.

10652 [Channel %1:] Axis %2 gantry warning threshold exceeded

Parameters: %1 = Channel number
%2 = Axis

Definitions: The gantry following axis has exceeded the warning limit specified in MD37110 $MA_GANTRY_POS_TOL_WARNING.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
1. Check axis (uneven mechanical movement?)
2. MD not set correctly (MD37110 $MA_GANTRY_POS_TOL_WARNING). Changes to this MD take effect after a RESET.

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

10653 [Channel %1:] Axis %2 gantry error threshold exceeded

Parameters: %1 = Channel number
%2 = Axis

Definitions: The gantry following axis has exceeded the error limit (actual value tolerance) specified in MD37120 $MA_GANTRY_POS_TOL_ERROR.

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.
1. Check axis (uneven mechanical movement?)
2. MD not set correctly (MD37120 $MA_GANTRY_POS_TOL_ERROR). A POWER ON is necessary after modifying the MD.
If the axes are not yet referenced, MD37130 $MA_GANTRY_POS_TOL_REF is the trigger condition for the error message.

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

10654 [Channel %1:] Waiting for synchronization start of gantry group %2

Parameters: %1 = Channel number
%2 = Gantry unit

Definitions: The alarm message appears when the axes are ready for synchronization. The gantry grouping can now be synchronized. The actual value difference between the master and slave axes is greater than the gantry warning limit MD 37110 $MA_GANTRY_POS_TOL_WARNING. The synchronization must be started explicitly with the NC/PLC interface signal <StartSynchronisationGantry/> (Start gantry synchronization).

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
See Function Manual, Special Functions, Gantry Axes (G1)

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

10655 [Channel %1:] Synchronization of gantry group %2 in progress

Parameters: %1 = Channel number
%2 = Gantry unit

Definitions: No further explanation.

Reaction: Alarm display.

Remedy: --

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

10656 [Channel %1:] Axis %2 gantry slave axis dynamically overloaded

Parameters: %1 = Channel number
%2 = Axis

Definitions: The indicated gantry slave axis is dynamically overloaded, i.e. the slave axis cannot follow the master axis dynamically

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

Remedy: Please inform the authorized personnel/service department. Compare the axial machine data of the gantry slave axis with the data of the gantry master axis

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

10657 [Channel %1:] Axis %2 power OFF in the gantry error limit exceeded status

Parameters: %1 = Channel number
%2 = Axis

Definitions: Gantry error limit exceeded status (alarm 10653) has been switched off.
The error can only be removed by deleting MD37135 $MA_GANTRY_ACT_POS_TOL_ERROR or by deactivating the extended monitoring (MD37150 $MA_GANTRY_FUNCTION_MASK Bit0).

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.
1. Remove a mechanical misalignment
2. Check axis (uneven mechanical movement?)
3. Delete MD37135 $MA_GANTRY_ACT_POS_TOL_ERROR or deactivate the extended monitoring
4. MD37120 $MA_GANTRY_POS_TOL_ERROR is set incorrectly
   If the MD is changed, a Power ON will be required.

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

10658 [Channel %1:] Axis %2 impermissible axis status %3.

Parameters: %1 = Channel number
%2 = Axis number
%3 = Error ID and gantry unit.

Definitions: Error ID and gantry unit
- 30XX => Gantry group cannot be closed, as not all gantry axes are in one channel.
- 40XX => Gantry group cannot be closed, as the gantry axes have different axis states, for example the axis is assigned to the PLC.
- 50XX => Gantry group is to change channel due to a PLC request, not all gantry axes are known in the new channel.
- 60XX => Gantry group is to be transferred to the channel due to an NC program request, but the channel does not know all the gantry axes.

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

Remedy: Error ID:
- 30XX => assign all gantry axes to the current channel, for example via axis exchange.
- 40XX => set all axes of the gantry group to the same axis state, for example assign all axes to the NC program, or assign all axes to the PLC.
- 50XX => make all gantry axes known to the required channel.
- 60XX => make all gantry axes known to the required channel.
:end

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: The path programmed for the axis violates the currently valid software limit switch. The alarm is activated when preparing the part program block.
If bit 11=0 in the machine data MD11411 $MN_ENABLE_ALARM_MASK, this alarm is issued instead of alarm 10722. If bit 11 is set in the machine data MD11411 $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered for the software limit switch 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: Check and correct positions in the NC program.
Please inform the authorized personnel/service department.
Check machine data: MD36100 $MA_POS_LIMIT_MINUS / MD36120 $MA_POS_LIMIT_MINUS2 and MD36110 $MA_POS_LIMIT_PLUS / MD36130 $MA_POS_LIMIT_PLUS2 for the software limit switches.

Check the axis-specific interface signals: V380x 1000.3 / .2 (2nd software limit switch plus/minus) to see whether the 2nd software limit switch is selected.
Check the currently active work offsets via the current frame.
Work offsets, overlaid movements ($AA_OFF), DRF and transformation components must also be checked.

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 motion planned for the axis violates the currently valid software limit switch.
The alarm is activated during the preprocessing of approach or rest blocks for REPOS.
Depending on MD11411 $MN_ENABLE_ALARM_MASK, bit11=0 this alarm is output instead of alarm 10723. If this MD11411 $MN_ENABLE_ALARM_MASK, bit11 is set in this machine data $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered 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 Manual for alarm 10723.

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 axis-specific NC/PLC interface signals V380x 1000.3 / .2 (2nd software limit switch plus/minus) 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 violates the working area limitation.
If bit 11=0 in machine data MD11411$MN_ENABLE_ALARM_MASK, this alarm is issued instead of alarm 10732. If bit 11 is set in machine dataMD11411 $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered for the software limit switch 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.

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: The motion planned for the axis violates the currently active working area limit.
The alarm is activated during the preparation of approach or rest blocks for REPOS.
This alarm is issued instead of alarm10733 if bit11 is not set in MD11411 $MN_ENABLE_ALARM_MASK.

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: It is not allowed to program more blocks than specified by MD20202 $MC_WAB_MAXNUM_DUMMY_BLOCKS between the WAB block and the block determining the approach and 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.

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 WAB 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 WAB block, the parameter DISR has not been specified or its value is less than or equal to 0.
During approach or retraction with circle and active tool radius, the radius of the internally generated - WAB contour is negative. The internally generated WAB contour is a circle with a radius which, when offset with the current offset radius (sum of tool radius and offset value OFFN), yields the tool center point path with the programmed radius DISR.

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 WAB motion before a previously activated WAB 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 block with travel information follows the approach block.
Before a retraction block, no block with travel information has been programmed in a program.
The tangent to be used for WAB motion is vertical to the current machining plane.

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.

10745 [Channel %1:] Block %2 WAB end position not clear

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

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

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 WAB block or the following block, or program a position in the machining plane in the WAB block as well.

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

10746 [Channel %1:] Block %2 block search stop for WAB

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

Definitions: A preprocessing stop has been inserted between an SAR approach block and the following block defining the tangent direction or between an SAR retraction 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 WAB retraction block with quarter circle or semi-circle (G248 or G348), the end point in the machining plane was not programmed, and either G143 or G140 without tool radius compensation is active.

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 changes are possible:
- Indicate end point in the machining plane in the WAB block.
- Activate tool radius compensation (effective for G140 only, not for G143).
- State retraction side explicitly with G141 or G142.
- Perform retraction with a straight line instead of a circle.

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

10748 [Channel %1:] Block %2 illegal retract plane with WAB

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

Definitions: By means of DISRP a position of the retraction plane has been programmed which is not situated between the safety distance (DISCL) and the starting point (during approach) and/or end point (during retraction) of the WAB movement.

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 cutter radius compensation (CRC) is allowed for if function G41 or G42 is programmed. The correction values are contained in parameter P6 (geometry value) and P15 (wear value) of the active correction data block Dx.

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

Remedy: Before calling the CRC with G41/G42, program a tool number under the address T...

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

10751 [Channel %1:] Block %2 danger of collision due to tool radius compensation

Parameters: %1 = Channel number

%2 = Block number, label

Definitions: The "Bottleneck detection" (calculation of intersection for the following compensated traversing blocks) has not been able to calculate a point of intersection for the reviewed number of traversing blocks. It is therefore possible that one of the equidistant paths violates the workpiece contour.

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

Remedy: Please inform the authorized personnel/service department. Check the part program and, if possible, modify the programming so that inside corners with smaller paths than the correction value are avoided. (Outside corners are not critical because the equidistants are lengthened or intermediate blocks are inserted, so that there is always a point of intersection).
Increase the number of reviewed traversing blocks via machine data MD20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS (default: 3), this increases the amount of calculation and consequently also the block cycle time.

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

10752 [Channel %1:] Block %2 overflow of local block buffer with tool radius compensation

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

Definitions: The cutter radius compensation must buffer a variable number of intermediate blocks in order to enable calculation of the equidistant tool path for each NC block. The size of the buffer cannot be determined by simple means. It depends on the number of blocks without traversing information in the compensation plane, the number of contour elements to be inserted and the shape of the curvature in spline and polynomial interpolation.
The size of the buffer is fixed by the system and cannot be changed via the MDs.

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

Remedy: Please inform the authorized personnel/service department.
Reduce the size of the buffer that has been assigned by modifying the NC program.
- By avoiding:
- Blocks without traversing information in the compensation plane
- Blocks with contour elements having a variable curvature (e.g. ellipses) and with curvature radii that are smaller than the compensation radius. (Such blocks are divided up into several subblocks).
- Reduce the number of reviewed blocks for collision monitoring (MD20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS).

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

10753 [Channel %1:] Block %2 selection of the tool radius compensation only possible in linear block

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

Definitions: Selection of 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.

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 possibilities are available:
Selection in the previous block
Insert intermediate block
Select approach behavior NORM

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

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 possibilities are available:
Deselection in the next block
Insert intermediate block
Select retract behavior NORM

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

10757 [Channel %1:] Block %2 changing the compensation plane while tool radius compensation is active not possible

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

Definitions: In order to change the compensation plane (G17, G18 or G19) it is first necessary to deselect the cutter radius compensation with G40.

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.

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 permissible number of empty blocks is limited by a machine data.

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
- Modify machine data
- Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.

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

10763 [Channel %1:] Block %2 path component of the block in the compensation plane becomes zero

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

Definitions: Due to 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.
The alarm can be suppressed with MD11410 $MN_SUPPRESS_ALARM_MASK bit1 = 1.

Reaction: Alarm display.

Remedy: - The behavior is correct at narrow locations that cannot be machined with the active tool.
- Modify the part program if necessary.
- Use tool with smaller radius if necessary.
- Program CDOF/CDOF2.

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. This situation can occur, for example, when a geometry axis is traversed between two positions as a positioning axis or when, with an active kinematic transformation (e.g. 5-axis transformation) the tool length compensation is altered.

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.

10776 [Channel %1:] Block%2 axis %3 must be geometry axis if tool radius compensation is active

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

Definitions: This alarm occurs when an axis that is required for tool radius compensation is not a geometry axis. With CUT2DF, the axis can be a positioning axis perpendicular to the machining plane; with all other types of compensation (CUT2DF, CUT3DC, CUT3DF, CUT3DFF), all geometry axes must be operated as such.

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.
On selection of G41/42, the axes involved must be known as GEOAX in the channel. It is possible by programming GEOAX() or G91 G0 X0 Y0 in the block prior to G41/42.

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

10777 [Channel %1:] Block %2 tool radius compensation: too many blocks with suppression of compensation

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

Definitions: The maximum permissible number of blocks with active compensation suppression with tool radius compensation is limited by MD20252 $MC_CUTCOM_MAXNUM_SUPPR_BLOCKS.

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.
- Modify machine data.
- Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.

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

10778 [Channel %1:] Block %2 preprocessing stop with active tool radius compensation

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

Definitions: If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the SD42480 $SC_STOP_CUTCOM_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).

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

Remedy: - Continue machining with CANCEL and Start.
- Modify part program.
- Set SD42480 $SC_STOP_CUTCOM_STOPRE to FALSE.

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: A tool change on which the edge offset (difference between edge center and edge reference point) changes, is only permissible in straight and polynomial blocks.
It is impermissible in circular blocks, involute blocks and in blocks including rational polynomials with maximum permissible numerator and denominator degrees.

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.
- Set SD42480 $SC_STOP_CUTCOM_STOPRE to FALSE.

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

10784 [Channel %1:] Block %2 illegal tool for tool radius compensation with constraint surface

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

Definitions: When activating the tool radius compensation with constraint surface, an illegal tool type is active.
Only cutting tools of the tool types 1 to 399 are admitted with the following exceptions:
- 111 ball end milling cutter
- 155 torus milling cutter
- 156 torus milling cutter
- 157 torus milling cutter

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

Remedy: Use another tool.

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

10790 [Channel %1:] Block %2 plane change during linear programming with angles

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

Definitions: The active plane was changed between the first and second subblock when programming two straight lines with angle parameters.

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.

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.

10800 [Channel %1:] Block %3 axis %2 is not a geometry axis

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

Definitions: With an active transformation or a frame with a rotation component the geometry axes are needed for block preparation. If a geometry axis has previously been traversed as positioning axis, it retains its status of "positioning axis" until it is again programmed as a geometry axis.
Because of the POSA motion beyond block boundaries, it is not possible to identify in the preprocessing run whether the axis has already reached its target position when the block is executed. This is, however, an unconditional requirement for calculating the ROT component of the frame or of the transformation.
If geometry axes are used as positioning axes, then:
1. No rotation may be specified in the current overall frame.
2. No transformation may be selected.

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 $MC_SPIND_DEF_MASTER_SPIND is available for the default or the keyword SETMS in the part program, thus allowing each spindle of the channel to be redefined as master spindle.

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

Remedy: Preset the master spindle with MD20090 $MC_SPIND_DEF_MASTER_SPIND[n]=m (n ... channel index, m ... spindel no.) or define it with an identifier in an NC part program before a G function that requires a master spindle is programmed.
The machine axis that is to be operated as a spindle must be equipped in MD35000 $MA_SPIND_ASSIGN_TO_MACHAX[n]=m (n ... machine axis index, m ... spindle no.) with a spindle number. Additionally, the MD20070 $MC_AXCONF_MACHAX_USED[n]=m (n ... channel axis index, m ... machine axis index) must be used to assign it to a channel (channel axis index 1 or 2).

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

10820 [Channel %1:] Rotary axis/spindle %2 not defined

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

Definitions: Revolutional feed has been programmed for contouring and synchronous axes or for an axis/spindle. However, the rotary axis/spindle from which the feed is to be deduced is not available.

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

Remedy: Correct part program or set the SD43300 $SA_ASSIGN_FEED_PER_REV_SOURCE correctly.

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

10860 [Channel %1:] Block %2 feedrate not programmed

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

Definitions: Cause:
A traversing velocity has not been programmed for the displayed traversing block.
Feed F or FZ:
With the traversing velocity defined by feed F or FZ, F or FZ was not reprogrammed after the feed type changed, for example linear feed G94 after revolutional feedrate G95 F or G95 FZ.
Modal feed FRCM:
With modal traversing velocity FRCM defined for rounding RND or chamfering CHF, feed FRCM was not reprogrammed after the feed type changed, for example linear feed G94 after revolutional feedrate G95, or revolutional feedrate G95 F after tooth feedrate G95 FZ.
Note:
Feed FRCM also has to be reprogrammed when the feed type changes if the current traversing block does not contain chamfering CHF or rounding RND, but the feed FRCM was programmed active, that is unequal to 0, before the feed type changed.

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 [rev/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] or under the address FZ in [mm/tooth].
- 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 MD32060 $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: Modify the program so that no reference is possible to the program itself.

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

10870 [Channel %1:] Block %2 facing axis for constant velocity not defined

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

Definitions: Constant cutting speed was selected although no transverse axis was applied as reference axis for constant cutting speed or assigned through SCC[AX].
Constant cutting speed can be activated as follows:
- Basic position G96, G961 or G962 of G group 29 during booting
- Programming of G96, G961 or G962

A reference axis for G96, G961 or G962 can be applied as a transverse axis in MD20100 $MC_DIAMETER_AX_DEF or defined through the instruction SCC[AX].

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

Remedy: Please inform the authorized personnel/service department. Check MD20100 $MC_DIAMETER_AX_DEF. Before programming G96, G961 or G962 a transverse axis must be defined as a reference axis for constant cutting speed via MD20100 $MC_DIAMETER_AX_DEF or SCC[AX].

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

10880 [Channel %1:] Block %2 too many empty blocks between two traversing blocks when inserting chamfers or radii

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

Definitions: Between 2 blocks containing contour elements and which are to be joined with a chamfer or a radius (CHF, RND), more blocks without contour information have been programmed than provided for in the MD20200 $MC_CHFRND_MAXNUM_DUMMY_BLOCKS.

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

Remedy: Please inform the authorized personnel/service department. Modify the part program in order that the permissible number of dummy blocks is not exceeded or adapt the channel-specific machine data MD20200 $MC_CHFRND_MAXNUM_DUMMY_BLOCKS (dummy blocks with chamfers/radii) to the maximum number of dummy blocks.

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

10881 [Channel %1:] Block %2 overflow of local block buffer in the case of 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 part program such that the number of dummy blocks is reduced.

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

10882 [Channel %1:] Block %2 activation of chamfers or radii (non-modal) without traversing movement in the block

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

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 the plane
A plane change has taken place
The permissible number of empty blocks without traversing information (dummy blocks) has been exceeded.

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

Remedy: Please inform the authorized personnel/service department. Correct the part program according to the above error description or change the number of dummy blocks in the channel-specific MD20200 $MC_CHFRND_MAXNUM_DUMMY_BLOCKS to comply with the maximum number allowed for in the program.

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

10883 [Channel %1:] Block %2 chamfer or fillet has to be reduced

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

Definitions: This alarm is output, if at least one of the relevant blocks when inserting chamfers or radii is so short, that the contour element to be inserted must be reduced against its originally programmed value. The alarm occurs only if bit 4 is set in the MD11411 $MN_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: When the path axis waveforms were analyzed during block preparation, a large local deviation relative to the path velocity was detected in the velocity waveform of one or more path axes.
Such a situation can have the following causes:
- The path runs close to singular positions of the machine kinematics.
- The programmed contour characteristic is very uneven.
- The FGROUP definition is unfavorable relative to the contour.
- The setting MD28530 $MC_MM_PATH_VELO_SEGMENTS=0 is inadequate for curvature changes occurring within one block. This problem occurs more frequently with G643, G644 and COMPCAD.
- A kinematic transformation has been implemented with insufficient numerical accuracy.
The path velocity is generally reduced substantially in order to avoid axis overloads safely. An apparent machine standstill may occur. Severe axis movements occur suddenly as soon as the singular position is reached.

Reaction: Local alarm reaction.
Alarm display.

Remedy: Dividing a block into several smaller ones often provides an improvement.
If MD28530 $MC_MM_PATH_VELO_SEGMENTS=0 is set, then the alarm may be avoidable by a value MD28530 $MC_MM_PATH_VELO_SEGMENTS=3 or 5, as the blocks are then analyzed considerably more accurately.

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

10912 [Channel %1:] Block %2 preprocessing and main run might not be synchronized

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

Definitions: The preset positioning axis run cannot be accurately calculated beforehand. The reason for this is either that the axes involved in the transformation are traversed as positioning axes or that a transformation pole is circumnavigated too frequently by the curve.
The velocity check is performed starting from this block in the main run. It is more conservative than with anticipated calculation. The LookAhead function is deactivated. If it is not possible to take over the velocity check into the main run, part program processing is aborted.

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 Look Ahead function remains deactivated until preprocessing can be based on defined conditions again (e.g. as a result of change from JOG->AUTO, tool or tool edge change).

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

10913 [Channel %1:] Block %2 negative feed profile is ignored

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

Definitions: The given feed profile is in part negative. However, negative path feed is not allowed. The feed profile is ignored. The specified feed block end value is taken when traversing over the entire block.

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 [Channel %1:] Block %2: movement not possible while transformation active.

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

Definitions: The machine kinematics does not allow the specified motion. Transformation-dependent error causes can be in: TRANSMIT: A (circular) area exists around the pole, where positioning is not possible. The area is caused by the fact that the tool reference point cannot be traversed as far as into the pole. The area is defined by:
- the machine data (MD24920 $MC_TRANSMIT_BASE_TOOL..)
- the active tool length compensation (see $TC_DP..). Whether the tool length compensation is included in the calculation depends on the working plane selected (see G17,..).
- The machine stops before the faulty block.

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

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

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

10930 [Channel %1:] Block %2 interpolation type not allowed in stock removal contour

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

Definitions: The following types of interpolation are allowed in the contour program for stock removal: G00, G01, G02, G03, CIP, CT

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

Remedy: In the contour subroutine, program only path elements that consist of straight lines and arcs.

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

10931 [Channel %1:] Block %2 incorrect stock removal contour

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

Definitions: The following errors occurred in the subroutine for the contour during stock removal:
- Full circle
- Overlapping contour elements
- Wrong start position

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

Remedy: The errors listed above must be corrected in the subroutine for the stock removal contour.

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

10932 [Channel %1:] Block %2 preparation of contour has been restarted

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

Definitions: The first contour preparation/contour decoding run must be terminated with EXECUTE.

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

Remedy: Program the keyword EXECUTE to terminate the contour preparation in the part program before again calling up contour segmentation (keyword CONTPRON).

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

10933 [Channel %1:] Block %2 contour programm does not contain enough contour blocks

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

Definitions: The contour program contains:
- Less than 3 contour blocks with CONTPRON
- No contour blocks with CONTDCON

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

10934 [Channel %1:] Block %2 array for contour segmentation is set too small

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

Definitions: During contour segmentation (activated with the keyword CONTPRON), the field for the contour table has been detected as too small. For every permissible contour element (circle or straight line) there must be a row in the contour table.

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

Remedy: Base the definition of the field variables of the contour table on the contour elements to be expected. The contour segmentation divides some NC blocks into as many as 3 machining cuts. Example: N100 DEF TABNAME_1 [30, 11] Field variables for the contour table provide for 30 machining cuts. The number of columns (11) is a fixed quantity.

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

10962 [Channel %1:] Block %2 function %3 not possible with path correction

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

Definitions: With this software release, the specified function can not yet be used together with tool radius compensation. Please modifiy 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 the address

Definitions: Most addresses (address types) may only be programmed once in an NC block, so that the block information remains unambiguous (e.g. X... T... F... etc. - exception: G and M functions).

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 from the NC program addresses that occur more than once (except for those where multiple value assignments are allowed).
- Check whether the address (e.g. the axis name) is specified via a user-defined variable (this may not be easy to see if allocation of the axis name to the variable is performed in the program through computational operations only).

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

12010 [Channel %1:] Block %2 address %3 address type programmed too often

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

Definitions: The number of times each address type may occur in a NC block is defined internally (for instance, all axes together form one address type to 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', 'CIC', 'CAC', 'ACN', 'ACP', 'CACN', 'CACP'. Not each of these address modifications can be used for each address type. The Programming Guide specifies which of these can be used for the various address types. If this address modification is applied to address types that are not allowed, then the alarm is generated, e.g.:
N10 G02 X50 Y60 I=DC(20) J30 F100
interpolation parameters with DC.

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 is then positioned on the incorrect block.
Apply non-modal address modifications only for permissible addresses, in accordance with the Programming Manual.

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

12040 [Channel %1:] Block %2 expression %3 is not of data type 'AXIS'

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

Definitions: Some keywords require that the data in their parameters be written in variables of the type "AXIS". For example, in the keyword PO the axis identifier must be specified in the parenthesized expression, and it must be defined as a variable of the AXIS type. With the following keywords only parameters of the AXIS type are possible:
AX[..], FA[..], FD[..], FL[..], IP[..], OVRA[..], PO[..], POS[..], POSA[..]
Example:
N5 DEF INT ZUSTELL=Z1 incorrect, this does not specify an axis identifier but the number 26 161
N5 DEF AXIS ZUSTELL=Z1 correct
:
N10 POLY PO[X]=(0.1,0.2,0.3) PO[Y]=(22,33,44) &PO[INFEED]=(1,2,3)

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 part program in accordance with the instructions given in the Programming Guide.

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

12050 [Channel %1:] Block %2 DIN address %3 not configured

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

Definitions: The name of the DIN address (e.g. X, U, X1) is not defined in the control. In addition to the fixed DIN addresses, the control also has variable addresses. Refer to "Variable addresses" in the Programming Guide. The names of these addresses can be altered by machine data.
e.g.: DIN identifier -> Configured identifier
G01 -> LINE, G04 -> WAIT ...

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 DIN block accordingly.

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

12060 [Channel %1:] Block %2 same G group programmed repeatedly

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

Definitions: The G functions that can be used in the part program are divided into groups that are syntax defining or non-syntax defining. Only one G function may be programmed from each G group. The functions within a group are mutually preclusive.
The alarm refers only to the non-syntax defining G functions. If several G functions from these groups are called in one NC block, the last of these in a group is active in each case (the previous ones are ignored).
Syntax defining G functions: 1. to 4th G group
Non-syntax defining G functions: 5. to nth G group

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

Definitions: At the text position shown, the grammar in the block is incorrect. The precise reason for this error cannot be specified in more detail because there are too many possibilities.
Example 1:
N10 IF GOTOF ... ; the condition for the jump is missing!
Example 2:
N10 DEF INT VARI=5
N11 X VARI ; the operation is missing for the X and VARI variables

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 = Disallowed parameters in the text

Definitions: The programmed function has been predefined; no parameters are allowed in its call. The first unexpected parameter is displayed.
Example: On calling the predefined subroutine TRAFOF (switching off a transformation) parameters have been transferred (one or more).

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 permissible together with the valid syntax defining G function, e.g. G1 I10 X20 Y30 F1000.
An interpolation parameter must not be programmed in the linear block.

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 block structure and correct in accordance with the programming requirements.

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

12120 [Channel %1:] Block %2 G function not separately programmed

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

Definitions: The G function programmed in this block must be alone in the block. No general addresses or synchronous actions may occur in the same block. These G functions are:
G25, G26: Working area and spindle speed limitation
G110, G111, G112: Pole programming with polar coordinates
G92: Spindle speed limitation with v constant
STARTFIFO, STOPFIFO: Control of preprocessing buffer
E.g. G4 F1000 M100: no M function allowed in the G4 block.

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 construct in the source text

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

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).
Example 1:
Arithmetic operation
N10 DEF INT OTTO
N11 DEF STRING[17] ANNA
N12 DEF INT MAX
:
N50 MAX = OTTO + ANNA
Example 2:
Value assignment
N10 DEF AXIS DRILL N11 DEF INT OTTO : N50 OTTO = DRILL

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.
Alter the definition of the variables used such that the required operations can be executed.

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

12160 [Channel %1:] Block %2 Value %3 lies beyond the value range

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

Definitions: The programmed constant or the variable lies beyond the value range that has previously been established by the definition of the data type.
An initial value in a DEF or REDEF instruction lies beyond the upper (ULI) or lower (LLI) limit values that have been programmed or already exist in the DEF instruction.

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 value of the constant or adapt data type. If the value for an integer constant is too great, it can be specified as real constant by adding a decimal point.
Example:
R1 = 9 876 543 210 Correct: R1 = 9 876 543 210.
Value range INTEGER: +/-(2**31 - 1)

Value range REAL: +/-(10**-300 .. 10**+300)

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

12170 [Channel %1:] Block %2 name %3 defined several times

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Symbol in block

Definitions: The symbol shown in the error message has already been defined in the active part program. Note that user-defined identifiers may occur more than once if the multiple definition occurs in other (sub)programs, i.e. local variables may be redefined with the same name if the program has been exited (subprograms) or has already been concluded. This applies both to user-defined symbols (labels, variables) and to machine data (axes, DIN addresses and G functions).

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 ERG = VARA - (- VARB) ; correct notation
N10 ERG = VARA - - VARB ; 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 array definition, with multi-dimensional arrays define a second 2-dimensional array if necessary and operate it with the same field index.

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

12200 [Channel %1:] Block %2 symbol %3 cannot be created

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

Definitions: The symbol to be created with the DEF instruction cannot be created because:
- it has already been defined (e.g. as variable or function)

- the internal memory location is no longer sufficient (e.g. with large arrays)

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

Remedy: Make the following checks:
- Check with the text editor whether the name to be allocated in the active program cycle (main program and called subprograms) has already been used.
- Estimate the memory requirements for the symbols already defined and reduce these if necessary by using fewer global and more local variables.

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

12210 [Channel %1:] Block %2 string %3 too long

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

Definitions: - In the definition of a variable of type STRING, it has been attempted to initialize more than 100 characters.
- In an allocation, it has been found that the string does not fit in the given variable.

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 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.

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 hexadecimal 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.

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. Example: DEF FRAME LOCFRAME = CTRANS(X,200)
Equally, no default values can be programmed for axes in the program run during field initialization via SET.
A REDEF instruction with PRLOC is only permitted for setting data,
but not for machine data or variables.

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

Remedy: IPerform initialization in separate block in the execution part of the program: DEF FRAME LOCFRAME LOCFRAME = CTRANS(X,200)
When using for axis variables:
Replace DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = SET (X, , Y) by: DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = X AXIS_VAR [7] = Y
If REDEF ... INIRE, INIPO, INICF, PRLOC changes the behavior of a GUD, LUD etc., then the MD11270 $MN_DEFAULT_VALUES_MEM_MASK must equal 1.

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
Keyword
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

Definitions: The string of instructions on the right side of the macro is limited to 256 characters. If an attempt is made to define a longer character string under one macro (possible only through V.24 input of NC blocks, because communication between operator panel and NCK is limited to a block length of 242 characters), an alarm is displayed.

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 unambiguous and may not be repeated in the control (exception: local variables).

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 Z missing!

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 a variable to all REF parameters (call-by-reference parameters) of the subroutine when calling.
No variable must be assigned to "normal" formal parameters (call-by-value parameters), as these are defaulted with 0.

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

12310 [Channel %1:] Block %2 axis parameter missing on procedure call %3

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

Definitions: When calling the subroutine, an AXIS parameter is missing which, according to the EXTERN declaration, should be present.
With the EXTERN instruction, user-defined subroutines (procedures) are made "known" that have a parameter transfer.
Procedures without parameter transfer require no EXTERN declaration.
Example:
Subroutine XYZ (with the formal parameters):
PROC XYZ (INT X, VAR INT Y, AXIS A, AXIS B)
EXTERN instruction (with variable types):
EXTERN XYZ (INT, VAR INT, AXIS, AXIS) Subroutine call (with actual parameters):
N10 XYZ (, Y1, R_TISCH)
Variable X is defaulted with value 0
Variable Y is supplied with the value of the variable Y1 and returns the results to the calling program after the subroutine run
Variable A is supplied with the axis in R_TISCH
Variable B missing!

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

Definitions: When calling a procedure (a subroutine) it is found that the type of the actual parameter cannot be converted into the type of the formal parameter. There are two possible cases:
- Call-by-reference parameter: Actual parameter and formal parameter must be of precisely the same type, e.g. STRING, STRING.
- Call-by-value parameter: Actual parameter and formal parameter can in principle be different providing conversion is basically possible. In the present case, however, the types are generally not compatible, e.g. STRING -> REAL.
Overview of type conversions:
- from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: -
- from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: -
- from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: -
- from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: -
- from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: -
- from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: -
- from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes
1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
2) String length 0 => FALSE, otherwise TRUE.
*) At type conversion from REAL to INT fractional values that are >=0.5 are rounded up, others are rounded down.

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 transfer parameters of the subroutine call and define the application accordingly as call-by-value or call-by-reference-parameter.

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

12340 [Channel %1:] Block %2 number of parameters too high %3

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

Definitions: When calling a function or a procedure (predefined or user-defined) more parameters were transferred than defined.
Predefined functions and procedures: The number of parameters has been set permanently in the NCK.
User-defined functions and procedures: The number of parameters is established by type and name in the definition.

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.

12360 [Channel %1:] Block %2 dimension of parameter %3 incorrect

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

Definitions: The following possibilities of error must be checked:
- The current parameter is an array, but the formal parameter is a variable
- The current parameter is a variable, but the formal parameter is an array
-

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 NC part program in accordance with the cause of error as listed above.

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

12370 [Channel %1:] Block %2 range of values %3 not permissible

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

Definitions: A variable has been initialized with a value range outside an initialization block. The definition of program-global variables is allowed only in special initialization blocks. These variables can be initialized with a value range.

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 keyword OF) or define the variable as a global variable in the initialization block and initialize it with a value range.

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

12380 [Channel %1:] Block %2 maximum memory capacity reached

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

Definitions: The data definitions in this block cannot be processed because the maximum available memory for creating the data has been filled, or because the data block cannot accommodate any further data. The alarm can also occur if several subroutine calls are executed in sequence and no block with an effect on the machine is generated (motion, dwell, M function).

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.
- If new macro definitions are to be introduced -> Increase MD18160 $MN_MM_NUM_USER_MACROS
- If new GUD definitions are to be introduced -> Check MD18150 $MN_MM_GUD_VALUES_MEM, MD18130 $MN_MM_NUM_GUD_NAMES_CHAN, MD18120 $MN_MM_NUM_GUD_NAMES_NCK
- If the error occurs while executing an NC program with LUD definitions or when using cycle programs (the parameters count as LUD variable of the cycle program), the following machine data must be checked:
MD28040 $MC_MM_LUD_VALUES_MEM,
MD18242 $MN_MM_MAX_SIZE_OF_LUD_VALUE,
MD18260 $MN_MM_LUD_HASH_TABLE_SIZE,
MD28020 $MC_MM_NUM_LUD_NAMES_TOTAL,
MD28010 $MC_MM_NUM_REORG_LUD_MODULES

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

12390 [Channel %1:] Block %2 initialization value %3 cannot be converted

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

Definitions: During initialization, a value has been assigned to a variable that does not correspond to the type of the variable, nor can it be converted to the data type of the variable.
Overview of type conversions:
- from REAL to REAL: no, INT: yes1), BOOL: yes, CHAR: yes2), STRING: -
- from INT to REAL: yes, INT: no, BOOL: yes, CHAR: yes2), STRING: -
- from BOOL to REAL: yes, INT: yes, BOOL: no, CHAR: yes, STRING: -
- from CHAR to REAL: yes, INT: yes, BOOL: yes, CHAR: no, STRING: yes
- from STRING to REAL: -, INT: -, BOOL: yes, CHAR: yes3), STRING: no
1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
2) String length 0 => FALSE, otherwise TRUE.
3) If only one character.
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.

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 variable type such that the initialization value can be assigned, or
- Select initialization value in accordance with the variable definition.

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

12400 [Channel %1:] Block %2 field %3 element does not exist

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

Definitions: The following causes are possible:
- Impermissible index list; an axis index is missing
- Array index does not match the definition of the variables
- An attempt was made to access a variable at array initialization via SET or REP; this attempt did not correspond to the standard access. Single character access, partial frame access, omitted indices not possible.
A non-existent 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 size of the array. The1st number indicates the number of elements in the 1st dimension (number of rows), the 2nd number indicates the number of elements in the 2nd dimension (number of columns).
An array with 2 rows and 3 columns must be defined by specifying [2,3].

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

12410 [Channel %1:] Block %2 incorrect index type for %3

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

Definitions: In assigning a value to an element of an array variable, the array index was specified in a way that is not allowed.
Only the following are allowed as array index (in square brackets):
- Axis identifier, provided the array variable was defined as data type FRAME.
- Integer values for all other data types.

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.

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. A check must be made to determine whether all parameters really have to be transferred. If so, the formal parameters can be reduced by using global variables or R parameters, or by grouping together parameters of the same type to form an array and transfer them in this form.

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

12450 [Channel %1:] Block %2 label defined twice

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

Definitions: The label of this block already exists.
If the NC program is compiled off-line, the entire program is compiled block for block. During this procedure all multiple labels are recognized; this is not always the case with 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 2nd time. Use the editor to search the part program where this label occurs for the 1st time, and change one of the names.

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

12460 [Channel %1:] Block %2 maximum number of symbols exceeded with %3

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

Definitions: The max. number of variable definitions (GUD, LUD), macro definitions, cycle programs and/or cycle parameters (PROC instruction) that the controller's data management system is able to handle has been exceeded.
If this alarm occurs in conjunction with alarm 15175, not enough memory for the preprocessing of the cycle program definitions is available (PROC instruction).
If this alarm occurs in conjunction with alarm 15180, then this alarm shows the name of the file (INI or DEF file) causing the error.
(For a list of names of INI files and their meaning -> please refer to alarm 6010)

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

Remedy: Generally reduce the number of symbols in the affected block (possibly by using the array technique or by using R parameters), or adapt the corresponding machine data (see below).
MD28020 $MC_MM_NUM_LUD_NAMES_TOTAL with error in LUD blocks (i.e. if more variable definitions were made in the active part program than allowed by the MD).
GUD data blocks can cause errors as part of the 'initial.ini download' process (e.g. in the case of a series start-up) or by selective activation via PI service _N_F_COPY (activate GUD via HMI dialog).
If alarm 15180 refers to a GUD definition file, then MD18120 $MN_MM_NUM_GUD_NAMES_NCK or MD18130 $MN_MM_NUM_GUD_NAMES_CHAN is set to a value that is too small.

Macros are loaded during POWER ON/NCK RESET or selectively via PI service _N_F_COPY (activate macro via HMI dialog). If alarm 15180 refers to a macro definition file, then MD18160 $MN_MM_NUM_USER_MACROS is set to a value that is too small.
Cycle program definitions (PROC instruction) are reloaded at each POWER ON/NCK RESET. In case of failure check parameter %3 to find out whether the name of the cycle program has caused the error - in this case, the value of MD18170 $MN_MM_NUM_MAX_FUNC_NAMES should be increased, or whether the name of a cycle call parameter has caused the error - in this case, the value of MD18180 $MN_MM_NUM_MAX_FUNC_PARAM should be increased.

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

12470 [Channel %1:] Block %2 G function %3 is unknown

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

Definitions: With indirectly programmed G functions, an invalid or non-allowed group number has been programmed. Allowed group number = 1. and 5 max. number of G groups. In the displayed block, a non-defined G function has been programmed. Only "real" G functions are checked, which begin with the address G, e.g. G555. "Named" G functions such as CSPLINE, BRISK etc. are interpreted as subroutine names.

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 (or introduced by an OEM). Remove G function from the part program or program function call in accordance with the machine manufacturer's programming guide.

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

12475 [Channel %1:] Block %2 invalid G function number %3 programmed

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

Definitions: A non-allowed G function number (parameter 3) has been programmed for a G group with indirect G code programming. Only the G function numbers indicated in the Programming Guide "Fundamentals", Section 12.3 "List of G functions/Path conditions" are allowed.

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.

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.

12490 [Channel %1:] Block %2 access permission level %3 is not valid

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

Definitions: The desired access authorization, programmed with the keyword REDEF, has not been set. The desired protection level is either beyond the permitted value range or the protection level change is not allowed.
The protection level may be changed only if:
1. The current protection level is equal to or higher than the level originally defined, and
2. The new protection level is to be below the level originally defined.
The higher numerical values represent the lower protection levels. The lower 4 levels (from 7 to 4) correspond to the keyswitch positions, and the upper 4 levels are associated with 4 passwords.

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.
- Use the REDEF instruction only in the INITIAL_INI block
- Using the operator panel, set the current protection level to at least the same level as that of the variable with the highest level
- Program protection level within the permissible value range
- Only program new protection levels that are lower than the old values

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

12500 [Channel %1:] Block %2 in this module %3 is not possible

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

Definitions: The displayed keyword may not be used in this type of block and at this location (all files in the NCK are designated as blocks).
Block types:
Program block
Contains a main program or subroutine
Data block
Contains macro or variable definitions and possibly an M, H or E function
Initialization block
Contains only selected language elements for data initialization

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 (keyword) with its parameters from this block and insert in the block provided for this purpose.

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

12510 [Channel %1:] Block %2 too many machine data %3

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

Definitions: In the part program, in the machine data file (..._TEA) and in the initialization file (..._INI), no more than 5 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.

12530 [Channel %1:] Block %2 invalid index for %3

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

Definitions: In macro definitions, an attempt was made to define a G function with more than 3 decades or an M function with more than 2 decades as identifier of the macro.
Example:
_N_UMAC_DEF DEFINE G4444 AS G01 G91 G1234
DEFINE M333 AS M03 M50 M99
:
M17

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.
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 256 characters. After editing, for example, several macros in the block or a multiple nesting, this limit can be 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.
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.
-
_N_SMAC_DEF
_N_MMAC_DEF
_N_UMAC_DEF
_N_SGUD_DEF
_N_MGUD_DEF
_N_UGUD_DEF
Variable: DEF statement is missing
Program: PROC declaration is missing

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

Remedy: Press the NC Stop key and select the function "Compensation block" with the softkey PROGRAM CORRECT. The cursor positions itself on the incorrect block.
- Correct the name used (writing error)
- Check definitions of variables, subroutines and macros
- Declare subroutine with EXTERN, load subroutine to SPF-Dir
- Check interface definition of subroutine
- Check options. See also MD10711 $MN_NC_LANGUAGE_CONFIGURATION.

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.

Remedy: - Correct the name used (writing error)
- $TC_DPCx, $TC_TPCx, $TC_MOPCx, $TC_MAPCx, $TC_MPPCx, $TC_DPCSx, $TC_TPCSx, $TC_MOPCSx, $TC_MAPCSx, $TC_MPPCSx; with x=1,...10
- These are the OEM parameters of the tools magazines, The corresponding machine data value is set to < 10, or the option 'TM OEM parameters' has not been set.
- Use correct parameter number, or - if the name cannot be changed - set machine data correction (see MD18096 $MN_MM_NUM_CC_TOA_PARAM, ... MD18206 $MN_MM_NUM_CCS_TOA_PARAM, ...).)
- Check the option (machine data are only effective when the option is enabled).

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

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 option (if MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 1) or the NC function (if MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 3)
related to this language command is not active.
But the name of the language command is known.
Each programming of this language command 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 "Compensation block" function by pressing the PROGRAM CORRECT softkey. The cursor positions itself on the incorrect block.
- Correct the name used (in the case of a typing error).
- Activate the NC function (if a language command of an inactive function has been programmed).
- Enable the option required (if a language command of a function with a non-enabled option has been programmed).
See also MD10711 $MN_NC_LANGUAGE_CONFIGURATION.

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 = Fine ID

Definitions: The identifier has not been defined for this system.

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 correction indicator will position in the incorrect block.
- Correct the name used (write error)
- Use a better software system in case of malfunction
- Check the definition of variables, subroutines and macros
- Declare a subroutine with EXTERNAL; load the subroutine to SPF-Dir
- Check the interface definition of the subroutine

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

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 is part of the NC language scope and therefore 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 key NC Stop and select "Correction block" function by pressing softkey "Program correct". The correction indicator is set to the incorrect block.
- Correct the name used (typing error)
- With MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 2 or 4, only those language commands are created, the option of which has been set
or the function of which is active.

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, e.g. INT ->REAL.
Value ranges of the various variable types:
- REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023)
- INT: Property: Integers with signs, value range: +/-(231-1)
- BOOL: Property: Truth value TRUE, FALSE, value range: 0,1
- CHAR: Property: 1 ASCII character, value range: 0-255
- STRING: Property: Character string (max. 100 values), value range: 0-255
- AXIS: Property: Axis addresses, value range: Axis names only
- FRAME: Property: Geometric information, value range: As for axis paths

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

12590 [Channel %1:] Block %2 global user data cannot be created

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

Definitions: The number of global user data blocks is defined in MD18118 $MN_MM_NUM_GUD_MODULES.
Here, _N_SGUD_DEF corresponds to block 1, _N_MGUD_DEF corresponds to block 2, _N_UGUD_DEF corresponds to block 3, _N_GUD4_DEF corresponds to block 4 etc.
In the directory _N_DEF_DIR there is a file with definitions for global user data, the block number of which is greater than the number of blocks given in the MD.
The alarm may, however, also be caused by value zero in one of MD18120 $MN_MM_NUM_GUD_NAMES_NCK, MD18130 $MN_MM_NUM_GUD_NAMES_CHAN and by the definition of a variable with NCK or CHAN in one of the GUD definition files.

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

Remedy: Please inform the authorized personnel/service department. Increase MD18118 $MN_MM_NUM_GUD_MODULES.
Or, if it already has the correct value, check whether MD18120 $MN_MM_NUM_GUD_NAMES_NCK (if a variable has been defined with attribute NCK) or MD18130 $MN_MM_NUM_GUD_NAMES_CHAN (if a variable has been defined with attribute CHAN) is not zero.

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").

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

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

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: Modify part program.

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

12701 [Channel %1:] Block %2 illegal interpolation type for contour definition active

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

Definitions: In one contour definition block, G01 is not active as interpolation function. In one contour definition block, the linear interpolation always has to be selected with G01. G00, G02, G03, G33 etc. are not permitted.

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

Remedy: Modify part program. Program linear interpolation G01.

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

12710 [Channel %1:] Block %2 illegal language element in external language mode

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

Definitions: The programmed language element is not allowed or unknown in external language mode. Only the language elements from Siemens mode which are used for subprogram calls (except for Lxx) and the language constructs for program repetition with REPEAT (UNTIL) are allowed.

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

Remedy: Modify part program.
Check that the language command is available in Siemens mode. Switch to Siemens mode with G290. Program the command in the next block and switch back to the external language mode in the following block.

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

12720 [Channel %1:] Block %2 program number for macro call (G65/G66) missing

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

Definitions: During macro call with G65/G66 no program number was defined. The program number must be programmed with address "P".

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.

12722 [Channel %1:] Block %2 multiple ISO_2/3 macro or cycle calls in the block

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

Definitions: A mixture of cycle and macro calls are programmed in a block, e.g. cycle calls with G81 - G89 together with an M macro in the block or a G65/G66 macro call together with M macros in the block.
G05, G08, G22, G23, G27, G28, G29, G30, G50.1, G51.1, G72.1, G72.2 functions (ISO mode) also execute subroutine calls. Only one macro or cycle call can appear in an NC block.

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

Remedy: Deactivate modal cycles or modal macro calls if one of the above mentioned G functions has been programmed.

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

12724 [Channel %1:] Block %2 no radius programmed for cylinder interpolation activation/deactivation

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

Definitions: When programming G07.1 (cylinder interpolation TRACYL), no cylinder radius has been programmed. Selection of the cylinder interpolation (TRACYL) with G07.1 C <cylinder radius> deselect with G07.1 C0. For "C" the name of the rotary axis defined in the TRACYL machine data has to be programmed.

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 SD42162 $SC_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 SD42162 $SC_EXTERN_DOUBLE_TURRET_DIST.

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

12730 [Channel %1:] Block %2 no valid transformation machine data parameterized

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

Definitions: The machine data MD24100 $MC_TRAFO_TYPE_1, MD24110 $MC_TRAFO_AXES_IN_1[1], MD24210 $MC_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 MD24100 $MC_TRAFO_TYPE_1 and the rotary axis number in MD24110 $MC_TRAFO_AXES_IN_1[1] or MD24210 $MC_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

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: Modify 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: Alarm 14000 is output in the following situations:
- Parts program was not terminated with M30, M02 or M17.
- Executing from external: Download was aborted (e.g. because HMI was switched off).

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

Remedy:
- End parts program with M30, M02 or M17 and start parts program.
- Executing from external: If the download for the selected program was aborted, the default program _N_MPF0 is automatically selected with RESET The selection of the user program must be repeated after that.

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 reloading from an external source) a part file 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

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: A subroutine call was aborted because the called subroutine could not be opened.
The subroutine call can be executed via
- subroutine designator
- CALL / PCALL / MCALL command
- SETINT command
- M/T function replacement
- event-driven program calls (PROG_EVENT)
- selection of a PLC ASUB via PI "_N_ASUP__" and/or FB-4
- calling a PLC ASUB via interrupt interface (FC-9)

There are various reasons for the alarm:
- the subroutine is not in the parts program memory the subroutine
- the subroutine is not in the search path (selected directory, _N_SPF_DIR or cycle directories _N_CUS_DIR, _N_CMA_DIR, _N_CST_DIR
- the subroutine has not been released or is being edited
- faulty absolute path name in subroutine call:
  Examples of complete path names: /_N_directoryName_DIR/_N_programmName_SPF or /_N_WKS_DIR/_N_wpdName_WPD/_N_programmName_SPF. directoryName: MPF, SPF, CUS, CMA, CST (predefined directories). wpdName: application-specific designator for workpiece directories (max. of 24 signs). programmName: Name of subroutine (max. of 24 signs)
- A reload buffer for executing from external was called as subroutine.

Note: Unknown designators (string) found in the parts program line by themselves, are interpreted as subroutine calls.

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

Alarm display.

Remedy: Ensure that the subroutine (alarm parameter %3)
- is available in the parts program memory
- has been released and is not being edited
- is available in the search path if not being called via an absolute path name.

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

14012 [Channel %1:] Block %2 maximum subroutine level exceeded

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

Definitions: The maximum nesting depth of 8 program levels has been exceeded.
Subroutines can be called from the main program, and these in turn may have a nesting depth of 7.
In interrupt routines the maximum number of levels is 4!

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

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
%2 = Block number, label
%3 = Program name

Definitions: The selected parts program is not in the NCK memory or the access authorization for the program selection is from a higher level than the current control status.
During creation, this program received the protection level of the NC control which was active at the time.
In SW 5 or higher a program edited on HMI can no longer be started with NC Start.
The alarm will also be issued, if a file other than the specified definition file has been selected for the GUD or macro definition.

Reaction: Alarm display.

Remedy: Reload the program in the NCK memory or check and correct the name of the directory (workpiece overview) and the program (program overview) and reselect.

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

14015 [Channel %1:] Block %2 program %3 is not enabled

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

Definitions: The execution right currently set in the control (e.g. key switch position 0) is inadequate to execute part program %3.

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

Alarm display.

Remedy: - Raise the execution right to match the protection level of part program %3
- Assign a lower protection level to part program %3 or release (key switch protection level 0)

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

14016 [Channel %1:] Block %2 error when calling the subroutine via M/T function

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

Definitions: The following conflict was detected in a subprogram call per M or T function:
In the block referenced by parameter %2:
- An M or T function replacement has already been activated
- A modal subprogram call is active
- A subprogram return jump is programmed
- An end of program is programmed
- An M98 subprogram call is active (only in external language mode)
- T function replacement by D function programming in the same part program line is not possible with active TLC (G43/G44) in ISO2 system.

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 other program constructs. The part program must be corrected accordingly.

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

14017 [Channel %1:] Block %2 syntax error when calling the subroutine via M function

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

Definitions: When calling M code subroutine with parameter transfer, an illegal syntax was detected:
- Address extension not programmed as a constant.
- M function value not programmed as a constant.
Note:
If a parameter transfer has been programmed via MD10718 $MN_M_NO_FCT_CYCLE_PAR for an M function replacement, the following restriction applies to this M function: both the address extension and the M function value must be programmed for replacement as constants.

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 / current protection level

Definitions: To parts program command %3, a protection level has been assigned that is logically higher (smaller in 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 parts program. Please see the Siemens Progamming Guide or OEM documentation for the language commands permissible for the relevant system configuration.

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: Modify part program.

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

14021 [Channel %1:] Block %2 wrong value or wrong number of parameters on function or procedure call

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

Definitions: - An illegal parameter value was specified in a function or procedure call.
- An illegal number of actual parameters was programmed in a function or procedure call.

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

14040 [Channel %1:] Block %2 error in end point of circle

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

Definitions: In circular interpolation, either the circle radii for the initial point and the end point are further apart, or the circle center points are further apart, than specified in the machine data.
1. In circle radius programming the starting and end points are identical, thus the circle position is not determined by starting and end points.
2. Radii: The NCK calculates from the present start point and the other programmed circle parameters the radii for the start and the end point.
An alarm message is issued if the difference between the circle radii is either
- greater than the value in the MD21000 $MC_CIRCLE_ERROR_CONST (for small radii, if the programmed radius is smaller than the quotient of the machine data MD21000 $MC_CIRCLE_ERROR_CONST divided by MD21010 $MC_CIRCLE_ERROR_FACTOR), or
- greater than the programmed radius multiplied by the MD21000 $MC_CIRCLE_ERROR_FACTOR (for large radii, if the programmed radius is greater than the quotient of the machine data MD21000 $MC_CIRCLE_ERROR_CONST divided by MD21000 $MC_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 MD21000 $MC_CIRCLE_ERROR_CONST and MD21000 $MC_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: Modify part program.

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

14048 [Channel %1:] Block %2 wrong number of revolutions in circle programming

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

Definitions: In the circle programming, a negative number of full revolutions has been specified.

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

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. This may also occur with extensive expressions in synchronized actions.

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.

14060 [Channel %1:] Block %2 invalid skip level with differential block skip

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

Definitions: With "Differential block skip", a skip level greater than 7 has been specified. (In packet 1 specification of a value for the skip level is rejected by the converter as a syntax error, i.e. the only possibility is a "Suppress block" ON/OFF on one level).

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

Remedy: Enter a skip level (number behind the slash) less than 8.

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

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 MDI 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
%3 = Jump destination

Definitions: In conditional and unconditional jumps, the jump destination within the program must be a block with a label (symbolic name instead of block number). If no jump destination has been found with the given label when searching in the programmed direction, an alarm is output.
For parameterizable returns with RET to block number or label, the jump destination within the program must be a block with the block number or label (symbolic name instead of block number).
For returns over several levels (parameter 2), the jump destination must be a block within the program level you jumped to.
For returns with a string as return destination, the search string must be a name known in the control and the search string must be preceded in the block by a block number and/or a label only.

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

Remedy: Check NC part program for the following possible errors:
1. Check whether the target designation is identical with the label.
2. Is the jump direction correct?
3. Has the label been terminated with a colon?

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

14082 [Channel %1:] Block %2 label %3 program section not found

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Start or end label

Definitions: The start point for repetition of the program part with CALL <program name> BLOCK <start label> TO <end label> has not been found or the same program part repetition has been called recursively.

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

Remedy: Check the start and end labels for programming repetition in the user program.

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

14085 [Channel %1:] Block %2 instruction not allowed

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

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.

Remedy: Modify part program.

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

14088 [Channel %1:] Block %2 axis %3 doubtful position

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

Definitions: An axis position larger than 3.40e+38 increments has been programmed. This alarm can be suppressed with bit11 in MD11410 $MN_SUPPRESS_ALARM_MASK.

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.

14091 [Channel %1:] Block %2 illegal function, index %3

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

Definitions: A function was programmed or triggered which is not allowed in the current program context. The code of the function in question is encoded in the "index" parameter:
Index == 1: "RET" command was programmed in the main program level
Index == 2: Conflict between "Cancel level"/"Clear number of passes" and "Implicit GET"
Index == 3: Conflict ASUB start immediately after selection of overstore (up to P3)
Index == 4: MD10760 $MN_G53_TOOLCORR = 1 : SUPA/G153/G53 programmed in G75
Index == 5: POSRANGE command not programmed in synchronized action
Index == 6: SIRELAY command not programmed in synchronized action
Index == 7: GOTOF/GOTOB/GOTO command programmed with string variable in synchronized action.
Index == 8: COA application "cutting generator" not active

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

Remedy: Index == 1: Substitute "RET" command with M17/M30
Index == 2: Insert an auxiliary block (e.g. M99) after the subroutine call to which the "Cancel level"/"Clear number of passes" refers
Index == 3: Overstore an auxiliary block (e.g. M99), then start ASUB (up to P3)
Index == 4: With MD10760 $MN_G53_TOOLCORR = 1: Do not activate SUPA/G53/G153 in the G75 block
Index == 5: Program POSRANGE command in synchronized action

Index == 6: Program SIRELAY command in synchronized action
Index == 7: Program GOTOF/GOTOB/GOTO command with block number or label
Index == 8: Load COA application "cutting generator"

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

14092 [Channel %1:] Block %2 axis %3 is wrong axis type

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

Definitions: One of the following programming errors has occurred:
1. The keyword WAITP(x) "Wait with block change until the specified positioning axis has reached its end point" has been used for an axis that is not a positioning axis.
2. G74 "Reference point approach from the program" has been programmed for a spindle. (Only axis addresses are permitted).
3. The keyword POS/POSA has been used for a spindle. (The keywords SPOS and SPOSA must be programmed for spindle positioning).
4. If the alarm occurs with the "Rigid tapping" function (G331), the following causes are conceivable:
- The master spindle is not in position-controlled mode.
- Incorrect master spindle
- Master spindle without encoder
5. An axis name was programmed which no longer exists (e.g. when using axial variables as an index). Or NO_AXIS has been programmed as an index.
6. If 14092 is output as a note with alarm 20140 Motion-synchronous action: traversing of command axis, the following causes are possible:
- The axis is currently being traversed by the NC program already.
- An overlaid movement is active for the axis.
- The axis is active as following axis of a coupling.
- An interpolation compensation such as a temperature compensation is active for the axis.

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

Remedy:
- Correct the part program according to which of the above errors is involved.
- Program SPOS.
- Set the correct master spindle with SETMS.

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

14095 [Channel %1:] Block %2 radius for circle programming too small

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

Definitions: The radius entered for radius programming is too small, i.e. the programmed radius is smaller than half of the distance between start and end point.

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.

14096 [Channel %1:] Block %2 illegal type conversion

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

Definitions: During the program run, a variable value assignment or an arithmetic operation has caused data to be processed in such a way that they have to be converted to another type. This would lead to the value range being exceeded.
Value ranges of the various variable types:
- REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023)
- INT: Property: Integers with signs, value range: +/-(231-1)
- BOOL: Property: Truth value TRUE, FALSE, value range: 0,1
- CHAR: Property: 1 ASCII character, value range: 0-255
- STRING: Property: Character string (max. 100 values), value range: 0-255

- AXIS: Property: Axis addresses, value range: Axis names only
- FRAME: Property: Geometric information, value range: As for axis paths
Overview of type conversions:
- from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: -
- from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: -
- from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: -
- from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: -
- from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: -
- from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: -
- from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes
1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
2) String length 0 => FALSE, otherwise TRUE.
3) If only 1 character.
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.

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

Remedy: Modify the program section such that the value range is not exceeded, e.g. by a modified variable definition.

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

14097 [Channel %1:] Block %2 string cannot be converted to AXIS type

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

Definitions: The called function AXNAME - conversion of the transferred parameters of the STRING type to an axis name (return value) of the AXIS type - has not found this axis identifier in the machine data.

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

Remedy: Please inform the authorized personnel/service department. Check the transferred parameter (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:
MD10000 $MN_AXCONF_MACHAX_NAME_TAB
MD20060 $MC_AXCONF_GEOAX_NAME_TAB
MD20080 $MC_AXCONF_CHANAX_NAME_TAB

Select the transfer string in accordance with the axis name, and change the axis name in the machine data if necessary. (If a change of name is to take place via the NC part program, this change must first be validated by means of a "POWER-ON").

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

14098 [Channel %1:] Block %2 conversion error: no valid number found

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

Definitions: The string is not a valid INT or REAL number.

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

Remedy: Modify part program. If it is an entry, then you can check whether the string is a number via the preset function ISNUMBER (with the same parameter).

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

14099 [Channel %1:] Block %2 result in string concatenation too long

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

Definitions: The result of string chaining returns a string which is greater than the maximum string length laid down by the system.

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 query the size of the sum string before executing the chaining operation.

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

14130 [Channel %1:] Block %2 too many initialization values given

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

Definitions: On assigning an array by means of SET, more initialization values than existing array elements have been specified in the program run.

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

14160 [Channel %1:] Block %2 tool length selection without geometry axis specification

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

Definitions: If variant C (tool length acts on the programmed axis) is activated by machine data MD20380 $MC_TOOL_CORR_MODE_G43G44 for tool length compensation with H word and G43/G44 in ISO_2 mode, at least one geometry axis 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 MD20380 $MC_TOOL_CORR_MODE_G43G44 or the part program.

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

14165 [Channel %1:] Block %2 selected ISO H/D number %3 does not match tool %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = ISO H/D number
%4 = Tool number

Definitions: When an H or D number is programmed in ISO_2 or ISO_3 mode, it must be available in the active tool. The active tool may also be the last tool loaded on the master spindle or master toolholder. This alarm is output if there is no H or D number on this tool.

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

Remedy: Set ISO H/D number correctly.

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

14170 [Channel %1:] Block %2 illegal interpolation type with tool length compensation

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

Definitions: If tool compensation (G43/G44) is activated in language mode ISO_M, 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: Modify 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
%3 = H number of ISO mode

Definitions: The specified H number is not assigned to a 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: Modify 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
%3 = D number of ISO mode

Definitions: The specified D number is not assigned to a tool (language mode ISO_M).

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.

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: Modify part program.

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

14198 [Channel %1:] Block %2 illegal change of tool direction with tool offset

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

Definitions: If an offset is active in the tool direction, block change is not possible if this would change the assignment of the offset axes to the channel axes (plane change, tool change, cutter <=> turning tool, geometry axis exchange).

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.
- 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 MD20360 $MC_TOOL_PARAMETER_DEF_MASK is set) and bit 2 of this MD is also set, this tool may only be used in the plane active on tool selection. A plane change results in an alarm.

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.
Reset bit 2 in MD20360 $MC_TOOL_PARAMETER_DEF_MASK.

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

14200 [Channel %1:] Block %2 negative polar radius

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

Definitions: In the endpoint specification of a traversing block with G00, G01, G02 or G03 in polar coordinates, the polar radius entered for the keyword RP=... is negative.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

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

Remedy: Correct NC part program - permissible inputs for the pole radius are only positive absolute values that specify the distance between the current pole and the block end point. (The direction is defined by the polar angle AP=...).

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

14210 [Channel %1:] Block %2 polar angle too large

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

Definitions: In specifying the endpoints in a traversing block with G00, G01, G02 or G03 in polar coordinates, the value range of the polar angle programmed under the keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).

- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... referred to the last programmed point in the plane, G111 ... referred to the zero point of the current workpiece coordinate system (WCS), G112 ... referred to the last pole.

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.

14250 [Channel %1:] Block %2 negative pole radius

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

Definitions: In redefining the pole with G110, G111 or G112 in polar coordinates, the pole radius specified under keyword RP=... is negative. Only positive absolute values are permitted.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

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

Remedy: Correct the NC part program. Permissible inputs for the pole radius are only positive, absolute values that specify the distance between the reference point and the new pole. (The direction is defined with the pole angle AP=...).

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

14260 [Channel %1:] Block %2 pole angle too large

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

Definitions: In redefining the pole with G110, G111 or G112 in polar coordinates, the value range of the pole angle specified under keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

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:
- 1st For positioning axes:
- Handwheel override programmed for indexing axes,
- No position programmed,
- FA and FDA programmed for the same axis in the block.
- 2nd For contouring axes:
- No position programmed,
- G60 not active,
- 1st G group incorrect (only G01 to CIP).

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.

14320 [Channel %3:] Axis %4: handwheel %1 used twice (%2)

Parameters: %1 = Handwheel number
%2 = Use
%3 = Channel
%4 = Axis

Definitions: Informational alarm indicating that the mentioned handwheel is used twice:
The second parameter provides the explanation:
1: Block with axial handwheel override for this axis cannot be executed as the handwheel for this axis performs a DRF movement
2: Block with velocity override of the path cannot be executed as the handwheel performs a DRF movement for this axis of the path
3: Block with contour handwheel cannot be executed as the handwheel performs a DRF movement for this axis of the path
4: PLC axis with axial handwheel override cannot be started immediately as the handwheel performs a DRF movement for this axis
5: The axis is a reciprocating axis with axial handwheel override; the reciprocating movement cannot be started immediately as the handwheel performs a DRF movement for this axis
6: The DRF movement for this axis cannot be executed as an axial handwheel override is active for this axis with the handwheel
7: The DRF movement for this axis cannot be executed as a velocity override of the path with the handwheel is active and the axis belongs to the path
8: The DRF movement for this axis cannot be executed as the contour handwheel is active with this handwheel and the axis belongs to the path
9: The DRF movement for this axis cannot be executed as the axis is a PLC axis with handwheel override that is active with this handwheel

10: The DRF movement for this axis cannot be executed as the axis is active as reciprocating axis with handwheel override with this handwheel

Reaction: Alarm display.

Remedy: Use the handwheel for one purpose at a time only.

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.

14401 [Channel %1:] Block %2 transformation not available

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

Definitions: The required transformation is not available.

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 24... $MC_TRAFO_TYPE_... (assigns the transformation to part program instruction).

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 runs cannot be accurately calculated beforehand. Consequently, the position in the MCS is not known exactly. It might therefore be possible that a change in the multiple significance of the transformation has been performed in the main run although no provision was made for this in the preprocessing run.

Reaction: Alarm display.

Remedy: Modify part program. Synchronize preprocessing run and main run.

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

14404 [Channel %1:] Block %2 illegal parameterization of transformation

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

Definitions: Error has occurred when selecting transformation.
Possible causes of error:
- An axis traversed by the transformation has not been enabled:
- is being used by another channel (-> enable)
- is in spindle mode (-> enable with SPOS)
- is in POSA mode (-> enable with WAITP)
- is concurrent Pos axis (-> enable with WAITP)
- Parameterization via machine data has an error
- Axis or geometry axis assignment to the transformation has an error,
- Machine data has an error (-> modify machine data, restart)

Please note: Any axes not enabled might be signaled via alarm 14092 or alarm 1011 instead of alarm 14404.
Transformation-dependent error causes can be in: TRAORI: -
TRANSMIT:
- The current machine axis position is unsuitable for selection (e.g. selection in the pole) (-> change position slightly).
- Parameterization via machine data has an error.
- Special requirement with respect to the machine axis has not been fulfilled (e.g. rotary axis is not a modulo axis) (-> modify machine data, restart).
TRACYL:
The programmed parameter is not allowed when transformation is selected.
TRAANG:
- The programmed parameter is not allowed when transformation is selected.
- Parameterization via machine data has an error.
- Parameter is faulty (e.g. TRAANG: unfavorable angle value (-> modify machine data, restart)
Persistent transformation:
- Machine data for persistent transformation are wrong (-> consider dependencies, change machine data, restart)
Only with active "OEM transformation" compile cycle:
The axes included in the transformation must be referenced.

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.
Only with active "OEM transformation" compile cycle:
Reference the axes included in the transformation before selecting transformation.

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

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: Modify part program.

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

14412 [Channel %1:] Block %2 transformation active at geometry axis changeover

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

Definitions: It is not permissible to change the assignment of geometry axes to channel axes when transformation is active.

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.

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 part program and delete active tangential control with TANGDEL.

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 tangentially followed-up axis cannot be traversed as positioning axis.

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

Remedy: Change part program and delete active tangential control with TANGDEL.

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

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: For a tangential axis that is coupled during preparation, a rounding length must be indicated with TANGON() on activating the tangential control, or possibly occuring discontinuities of the tangential axis cannot be smoothed.

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.

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: Factor r as programmed on TLIFT for the relative lift-off path must be within range 0 =< r < 1.

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.

14500 [Channel %1:] Block %2 illegal DEF or PROC instruction in the part program

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

Definitions: NC part programs with high-level language elements are divided into a preceding definition part followed by a program part. The transition is not marked specifically; a definition statement is not allowed to follow the 1st 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. Conventional subroutine structure (without parameter transfer):
% SPF 123456
:
M17
2. Subroutine structure with keyword and subroutine name (without parameter transfer):
PROC UPNAME
:
M17
ENDPROC
3. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-value"):
PROC UPNAME (VARNAME1, VARNAME2, ...)
:
M17
ENDPROC
4. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-reference"):
PROC UPNAME (Typ1 VARNAME1, Typ2 VARNAME2, ...)
:
M17
ENDPROC

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

14520 [Channel %1:] Block %2 illegal PROC instruction in data definition section

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

Definitions: The PROC statement may only be programmed at the beginning of the subroutine.

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.

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
%3 = File name

Definitions: The download buffer for "execute from external" could not be created. Possible causes:
- Not enough memory available (for minimum see MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE)
- No resources available for HMI NCK communication (see MD18362 $MN_MM_EXT_PROG_NUM)
- The file already exists

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

Remedy:
- Release memory, e.g. by deleting part programs
- Modify MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE and/or MD18362 $MN_MM_EXT_PROG_NUM.

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

14601 [Channel %1:] Block %2 reload buffer could not be deleted

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

Definitions: The reload buffer for "execute from external" could not be deleted. Possible cause:
- HMI/PLC communication was not terminated.

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

Remedy: All reload buffers are cleared on POWER ON.

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

14602 [Channel %1:] Block %2 timeout while reloading from external.

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

Definitions: No connection could be made to the HMI while reloading external subprograms (EXTCALL) or executing from external drives) within the monitoring time set in MD10132 $MN_MMC_CMD_TIMEOUT.

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

Remedy:
- Check the connection to the HMI
- Increase MD10132 $MN_MMC_CMD_TIMEOUT.

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

14610 [Channel %1:] Block %2 compensation block not possible

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

Definitions: An alarm was output which could be eliminated basically via program correction. Since the error occurred in a program which is processed from external, a compensation block/program correction is not possible.

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

Alarm display.

Remedy: - Abort program with reset.
- Correct program on HMI or PC.
- Restart reloading (possibly with block search and interrupt location).

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

14700 [Channel %1:] Block %2 timeout during command to interpreter

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

Definitions: A timeout has occurred in control-internal commands such as ANWAHL (part program selection), RESET (channel reset), REORG (reorganization of the preprocessing buffer) and NEWCONFIG (change in the configuration-specific machine data = restart).

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 HMI area or in OEM application) error-free execution is possible on repeating the program or operator action.
Otherwise, the A&D MC system support should be contacted with a precise description of the error situation:
(contact SIEMENS AG, System Support for A&D MC products, Hotline (Tel.:see alarm 1000)

Program Continuation: Switch control OFF - ON.

14701 [Channel %1:] Block %2 number of available NC blocks reduced by %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of non-available blocks

Definitions: After reset, it has been found that the number of available blocks has decreased compared with the last reset. This is due to a system error. Part program execution can be resumed after the alarm has been acknowledged. If the number of blocks no longer available is less than MD28060 $MC_MM_IPO_BUFFER_SIZE, then the POWERON alarm 14700 is output.

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 = Identifier of the function which caused the error

Definitions: Initialization blocks are generated (or not) after control power-up, (program) RESET and (program) START, depending on the settings in machine data MD20110 $MC_RESET_MODE_MASK and MD20112 $MC_START_MODE_MASK.
Errors can occur because of incorrect machine data settings. The errors are output with the same error messages as would appear if the function had been incorrectly programmed in the part program.
This alarm is also generated in order to indicate that an error relates to the initialization sequence.
Parameter %3 specifies which function triggers the alarm:
Control power-up and (program) RESET:
Value:
0: Error during synchronization preprocessing/main run
1: Error on selection of tool length compensation
2: Error on selection of transformation
3: Error on selection of work offset

The macro definitions and cycle interfaces are also read in during the power-up procedure. If an error occurs here, this is indicated by value = 4, or value = 5
6: Error creating 2 1/2 D protection zones during power up.
(Program) START:
Value
100: Error during synchronization preprocessing/main run
101: Error on selection of tool length compensation
102: Error on selection of transformation
103: Error on selection of synchronized spindle
104: Error on selection of work offset
Particularly when tool management is active, it is possible that a tool on the spindle or the toolholder is disabled but still needs to be activated.
These tools are automatically activated on RESET. On START, machine data MD22562 $MC_TOOL_CHANGE_ERROR_MODE can be used to specify whether an alarm is to be generated or an automatic bypass strategy selected.
If the parameter contains 3 values from 200 to 203, this means that an insufficient number of NC blocks is available for NC block preparation on certain commands (ASUB start, overstore selection, teach-in).
Remedy: Increase machine data MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.

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

Remedy: Please inform the authorized personnel/service department.
If parameter %3= 0 -3:
If the alarm or alarms occur on RESET:
Check the settings of machine data MD20110 $MC_RESET_MODE_MASK,
MD20120 $MC_TOOL_RESET_VALUE, MD20121 $MC_TOOL_PRESEL_RESET_VALUE,
MD20122 $MC_TOOL_RESET_NAME (only if tool management is active),
MD20130 $MC_CUTTING_EDGE_RESET_VALUE, MD20132 $MC_SUMCORR_RESET_VALUE,
MD20126 $MC_TOOL_CARRIER_RESET_VALUE,
MD20150 $MC_GCODE_RESET_VALUES, MD20154 $MC_EXTERN_GCODE_RESET_VALUES,
MD20140 $MC_TRAFO_RESET_VALUE,
MD21330 $MC_COUPLE_RESET_MODE_1,
MD24002 $MC_CHBFRAME_RESET_MASK

If parameter %3= 100 - 104:
Check the setting of MD20112 $MC_START_MODE_MASK and the machine data specified under '..._RESET_...'. If tool management is active, if necessary remove the tool stated in the associated alarm from the toolholder/spindle and cancel the 'disabled' status.
If parameter %3= 4 or 5:
Check macro definitions in _N_DEF_DIR
Check cycle directories _N_CST_DIR and _N_CUS_DIR
If parameter %3= 6:
Alarm 18002 or 18003 was also issued. This alarm contains the number of the incorrectly defined protection zone and an identifier of what is incorrect about the protection zone. The system variables must be appropriately corrected.
If Parameter %3= 200 bis 203:
Increase MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.

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

14711 [Channel %1:] Transformation selection not possible as axis %2 not available

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

Definitions: Based on the configuration of machine data MD20110 $MC_RESET_MODE_MASK and MD20140 $MC_TRAFO_RESET_VALUE, a transformation shall be selected by performing a reset or control ramp-up. However, this is not possible as axis %2 required for this is not available. Possible reason: The axis was occupied by another channel or the PLC.

Reaction: Interface signals are set.

Alarm display.

Remedy: - Use the GET command to get axis %2 in the channel in which the transformation is to be selected.
- Select the transformation by means of the part program command.

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.

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.

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.

14762 [Channel %1:] Block %2 too many PLC variables programmed

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

Definitions: The number of programmed PLC variables has exceeded the maximum permissible number. The number is set in MD 28150 $MC_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 NCK system settings has been defined for all auxiliary functions by means of the MD11100 $MN_AUXFU_MAXNUM_GROUP_ASSIGN (default: 1)
For each user-defined auxiliary function to be assigned to a group, the assignment is effected through 4 channel-specific machine data.

Return jump from asynchronous subprogram with M02/M17/M30, whereby the M code is not alone in the block. This is impermissible if the asynchronous subprogram interrupts a block with WAITE, WAITM or WAITMC. Remedy: Program M02/M17/M30 alone in the block or replace via RET.
22010 AUXFU_ASSIGN_TYPE: type of auxiliary function, e.g. M
22000 AUXFU_ASSIGN_GROUP: required group
22020 AUXFU_ASSIGN_EXTENSION: any required extension
22030 AUXFU_ASSIGN_VALUE: function value

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.

14780 [Channel %1:] Block %2 unreleased option used (identification %3)

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

Definitions: A non-released option has been used in the block.
Identifier Brief description

1	LaserCtrl option
2	ClearCtrl option
3	FeedAdapt option
4	AaTOff option
5	Tang option
6	LeadCtab option
7	ELG option
8	Trafo5 option
9	Traoem option
10	Transmit option
11	Tracon option
12	Tracyl option
13	Traang option
14	Oscill option
15	SynSpi option
16	Repos option
17	Spline option
18	Involute option
19	Poly option
20	Compress option
23	Masl option
24	ExtLang or ExtLanguage option not activated
25	TechCycle option
26	Liftfast option
27	ProgAccel option
33	AllAsupSynact option
34	CmdAxSpind option
35	Mea2 option
36	ProgAnaOut option
37	OptAaTOff option
41	MachineMaintenance option
42	PathFeedSAInput option
45	ElecTransfer option
46	Cut3D option
47	CDA option
48	Reserved: generic coupling option
49	Measuring cycles option
50	ForceControl option

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 = Fine ID

Definitions: A non-active function is used in the block
Brief description of the identification
1 Transformation
2 H number of the tool

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

Remedy: - Modify part program.
- Activate function.

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.

14800 [Channel %1:] Block %2 programmed path speed less or equal to zero

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

Definitions: Zero or a negative F or FZ value has been programmed in conjunction with the G functions G93, G94, G95 or G96. The path velocity may be programmed in the range of 0.001 to 999 999.999 [mm/min, mm/rev, mm/tooth, deg/min, deg/rev] for the metric input system and 0.000 1 to 39 999.999 9 [inch/min, inch/rev, inch/tooth] 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. Values of 1 ... 200% are allowed.
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: Modify part program or machine data.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

14815 [Channel %1:] Block %2 negative thread pitch change programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: A negative thread pitch 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 keyword 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 keyword LIMS is modal and can either be placed in front of or within the block that selects the constant cutting speed.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

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: Modify part program.

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

14840 [Channel %1:] Block %2 incorrect value range for constant cutting speed

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

Definitions: The programmed cutting speed is not within the input range
Input range metric: 0.01 to 9 999.99 [m/min]
Input range inch: 0.1 to 99 999.99 [inch/min]

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.

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 guaranteeing that the dimensions are definitely taken over from the workpiece drawing (avoidance of calculation errors).

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

14910 [Channel %1:] Block %2 invalid angle of aperture for programmed circle

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

Definitions: When programming a circle by means of the opening angle, a negative opening angle or an opening angle greater than or equal to 360 degrees has been programmed.

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 (CIP) all 3 points (initial, end and intermediate points) are on a straight line and the intermediate point (programmed by means of interpolation parameters I, J, K) is not located between the initial and end points.
If the circle is the component of a helix, the specified number of turns (keyword TURN=...) determines further block processing:

- TURN>0: alarm display because the circle radius is infinitely great.
- TURN=0 and CIP specified between initial and end points. A straight line is generated between the initial and end points (without alarm message).

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 initial and end points of the circle or do not make use of this type of circle programming and instead program the circle with radius or opening angle or center point parameters.

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

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

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 part program, however:
1. Reduce log file size requirement by:
Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops STOPRE.
2. Increase the size of the logfile by means of the channel-specific machine data:
Modify MD28000 $MC_MM_REORG_LOG_FILE_MEM and
Modify MD 28010 $MC_MM_NUM_REORG_LUD_MODULES

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 part program, however:
1. Reduce log file size requirement by:

Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops (STOPRE).
2. Increase the size of the logfile by means of the channel-specific machine data:
Modify MD28000 $MC_MM_REORG_LOG_FILE_MEM and
Modify MD MD28010 $MC_MM_NUM_REORG_LUD_MODULES

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

15150 [Channel %1:] Block %2 reload from external aborted

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

Definitions: Execution from external was aborted because the reload buffer does not have enough machine function blocks (traversing blocks, auxiliary function, dwell time etc.). Background: When already executed machine function blocks are released, memory becomes available in the reload buffer. If machine function blocks are no longer released, nothing can be reloaded - this results in a deadlock situation. Example: Definition of extremely long curve tables via execution from external.

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

Remedy: Insert machine function blocks in the part progam.
- Increase the size of the reload buffer (MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE).
- Decrease the size of the curve table (Note: Blocks within CTABDEF/CTABEND are not machine function blocks).

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

15160 [Channel %1:] Block %2 wrong preprocessing configuration

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

Definitions: A block element is required, but the block element memory 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. Modify the block search configuration in MD28060 $MC_MM_IPO_BUFFER_SIZE (decrease size of IPO buffer if necessary) or MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.

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: Modify 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. In particular when loading new cycle program on the NCK, problems can occur if the value settings in MD18170 $MN_MM_NUM_MAX_FUNC_NAMES and MD18180 $MN_MM_NUM_MAX_FUNC_PARAM are too small.

Reaction: Alarm display.

Remedy: - Modify part program.
- If new cycle programs have been loaded on the NCK, you will normally need to increase the values of MD18170 $MN_MM_NUM_MAX_FUNC_NAMES and MD18180 $MN_MM_NUM_MAX_FUNC_PARAM. See also the explanations for alarm 6010.

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

15180 [Channel %1:] Block %2: Error on editing program %3 as INI/DEF file.

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

Definitions: Errors were found when processing an initialization program (INI file), or a GUD or macro definition file (DEF file).
The error message which is then displayed refers to the program specified here.

Reaction: Alarm display.

Remedy: Correct the initialization program (INI file), or the GUD or macro definition file (DEF file).
In connection with Alarm 12380 or 12460, also change the memory configuration.

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

15185 [Channel %1:] %2 errors in INI file

Parameters: %1 = Channel number
%2 = Number of detected errors

Definitions: Errors were found when processing initialization program _N_INITIAL_INI.
This alarm will also be output, if errors are found during editing of _N_INITIAL_INI in the GUD definition files or if errors are found on ramp-up in the macro definition files.

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

Remedy: Please inform the authorized personnel/service department. Correct the INI or DEF file or correct the MD and create a new INI file (via "Upload").

Program Continuation: Switch control OFF - ON.

15190 [Channel %1:] Block %2 not enough free memory for subroutine call

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

Definitions: The following deadlock has been found in the interpreter: Memory is needed for calling a subroutine. The module memory is, however, empty and there is no prospect of module memory becoming free again by executing the preprocessing/main run queue, because this queue is empty.

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

Remedy: Please inform the authorized personnel/service department. Increase machine data MD28010 $MC_MM_NUM_REORG_LUD_MODULES / MD28040 $MC_MM_LUD_VALUES_MEM / MD18210 $MN_MM_USER_MEM_DYNAMIC or program a preprocessing stop STOPRE before calling the subroutine.

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

15300 [Channel %1:] Block %2 invalid number-of-passed-blocks during block search

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

Definitions: In the function "Block search with calculation" a negative number of passes has been entered in column P (number of passes). The permissible range of values is P 1 - P 9 999.

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. The following block search orders are allowed.

Type	Meaning
1	Search for block number
2	Search for label
3	Search for string
4	Search for program name
5	Search for line number in a file

Reaction: Alarm display.

Remedy: Modify the block search command.

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

15330 [Channel %1:] Block %2 invalid block number as search target

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

Definitions: Syntax error! Positive integers are allowed as block numbers. Block numbers must be preceded by ":" and subblocks by an "N".

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.

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: The first axis programming after "search to block end point" is performed incrementally. This is not allowed in the following situations:
- After searching the target a transformation change has taken place.
- A frame with rotation component is active. The programmed axis is involved in the rotation.

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 SD42444
$SC_TARGET_BLOCK_INCR_PROG = 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 block does not exist

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

Definitions: The operator has selected an INI block for a read, write or execution function which:
1. Does not exist in the NCK range or
2. Does not have the necessary protection level required for performing the function.

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 NCK. The present protection level must be selected to be at least equal to (or greater than) the protection level that has been defined for the read, write or execution function at the time of creating the file.

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

15410 [Channel %1:] Block %2 initialization file contains invalid M function

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

Definitions: The only M function allowed in an Init block is the M02, M17 or M30 end-of-program function.

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) but no motion or synchronous actions.

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

15420 [Channel %1:] Block %2 instruction in current mode not allowed

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

Definitions: The alarm is output in the following situation:
The interpreter has detected an illegal instruction (e.g. a motion command) while processing an INI file or a definition file (macro or GUD).
In a GUD file, the access security for a machine data is to be changed with REDEF, although an ACCESS file (_N_SACCESS_DEF, _N_MACCESS_DEF, _N_UACCESS_DEF) is available.
Access rights for machine data can only be changed then via one of the ACCESS files with REDEF.

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

Remedy: - Correct the INI, GUD or macro file.
- Correct part program.

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

15460 [Channel %1:] Block %2 syntax error when locking

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

15800 [Channel %1:] Block %2 wrong starting conditions for CONTPRON/CONTDCON

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

Definitions: There is an error in the start conditions for CONTPRON/CONDCON:
- G40 not active
- SPLINE or POLY active
- Unknown machining type programmed
- Transferred machining direction not defined

- Definition of LUDs in incorrect subroutine level
- Transferred circle coordinates

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

15810 [Channel %1:] Block %2 wrong array dimension for CONTPRON/CONTDCON

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

Definitions: The number of columns for the array created for CONTPRON/CONTDCON does not conform to the current programming guide.

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

15900 [Channel %1:] Block %2 touch probe not allowed

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

Definitions: Measure with deletion of distance-to-go
In the part program, an illegal probe has been programmed with the command MEAS (measure with deletion of distance-to-go). The probe numbers
0 ... no probe
1 ... probe 1
2 ... probe 2
are allowed, whether the probe is actually connected or not.
Example:
N10 MEAS=2 G01 X100 Y200 Z300 F1000
Probe 2 with deletion of distance-to-go

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

Remedy: Include a probe number within the limits given above in the keyword MEAS=... This must correspond to the hardware connection of the probe.

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

15910 [Channel %1:] Block %2 touch probe not allowed

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

Definitions: Measure without deletion of distance-to-go
In the part program, an illegal probe has been programmed with the command MEAW (measure without distance-to-go). The probe numbers
0 ... no probe
1 ... probe 1
2 ... probe 2
are allowed, whether the probe is actually connected or not.
Example:
N10 MEAW=2 G01 X100 Y200 Z300 F1000
Probe 2 without deletion of distance-to-go

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

Remedy: Include a probe number within the limits given above in the keyword MEAW=... This must correspond to the hardware connection of the probe.

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

15950 [Channel %1:] Block %2 no traverse motion programmed

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

Definitions: Measure with deletion of distance-to-go
In the part program, no axis or a traversing path of zero has been programmed with the command MEAS (measure with deletion of distance-to-go).

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

Remedy: Correct the part program and add the axis address or the traversing path to the measurements block.

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

15960 [Channel %1:] Block %2 no traverse motion programmed

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

Definitions: Measure without deletion of distance-to-go
In the part program, no axis or a traversing path of zero has been programmed with the command MEAW (measure without deletion of distance-to-go).

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

Remedy: Correct the part program and add the axis address or the traversing path to the measurements block.

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

16020 [Channel %1:] Repositioning in block %2 is not possible

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

Definitions: Programming or operator action incorrect:
Repositioning via REPOS command is only possible in an asynchronous subprogram (interrupt routine).
If the REPOS command was programmed, e.g. in the main program or in a cycle, part program execution is aborted with alarm 16020.
In addition, the alarm is output in the following situations:
- Access to $AC_RETPOINT (repositioning point) outside an ASUP (e.g. in the main program)
- An axis to be repositioned was a oscillating axis with sychronous infeed (OSCILL) in the interrupted block and is now in a state that does not allow it to be traversed as a oscillating axis. Remedy: Change the axis to "neutral axis" state before repositioning with WAITP.
- An axis to be repositioned was an infeed axis for a oscillating axis in the interrupted block; now it can no longer be traversed as one. Remedy: Change the axis back to "POS axis" state before repositioning.

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

Remedy: Modify the part program if necessary.

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

16100 [Channel %1:] Block %2 spindle %3 not available in the channel

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

Definitions: Incorrect programming:
This channel does not recognize the spindle number.

The alarm can occur together with a dwell or a 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 MD35000 $MA_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 machine data MD20070 $MC_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 spindle by the spindle number converter. The alarm can be issued after improper use of SD42800 $SC_SPIND_ASSIGN_TAB[].

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

Remedy: Correct setting data or modify part program.

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

16111 [Channel %1:] Block %2 spindle %3 No speed programmed

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

Definitions: Programming of a speed is expected.

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

Remedy: Program speed with S[spindle number]=..

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

16200 [Channel %1:] Block %2 spline and polynominal interpolation not available

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

Definitions: The spline and polynomial interpolation are options that are not contained in the basic version of the control.

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

Remedy: Do not program spline and polynomial interpolation, or retrofit the necessary option.

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

16410 [Channel %1:] Block %2 axis %3 is not a geometry axis

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

Definitions: A geometry axis has been programmed that cannot be imaged on any machine axis in the current transformation (possibly there is no transformation active at the moment).
Example:
Without transformation: Polar coordinate system with X, Z, and C axis

With transformation: Cartesian coordinate system with X, Y, and Z, e.g. with TRANSMIT.

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

Remedy: Activate transformation type with TRAORI (n) or do not program geometry axes that do not participate in the transformation grouping.

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

16420 [Channel %1:] Block %2 axis %3 programmed repeatedly

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

Definitions: It is not allowed to program an axis more than once.

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:
Keyword ROT without further specification of axis and angle.
Example: N100 ROT

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

16440 [Channel %1:] Block %2 rotation programmed for non-existent geometry axis

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

Definitions: A rotation of a geometry axis which does not exist was programmed.

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.

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 keywords CHF= ..., RND=... or RNDM=...

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.

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 although no transverse axis with diameter programming has been applied.
Transverse axes can be applied with MD20100 $MC_DIAMETER_AX_DEF or MD30460 $MA_BASE_FUNCTION_MASK bit2 for diameter programming.
Diameter programming can be applied through:
- basic position DIAMON or DIAM90 of the G 29 group during booting
- programming of DIAMON or DIAM90
- programming of DIAMONA[AX], DIAM90A[AX] or DAC, DIC, RAC, RIC

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Please inform the authorized personnel/service department.
When programming DIAMON/DIAM90, a traverse axis must be configured via MD20100 $MC_DIAMETER_AX_DEF.
When programming DIAMONA[AX], DIAM90A[AX] or DAC, DIC, RAC, RIC, the AX axis must be a transverse axis for diameter programming configured via MD30460 $MA_BASE_FUNCTION_MASK bit2.
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.

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/SPOSA in front of the defective block in the part program.
Program Continuation: Clear alarm with NC START or RESET key and continue the program.

16720 [Channel %1:] Block %2 axis %3 thread pitch is zero

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

Definitions: No pitch was programmed in a thread block with G33 (thread with constant pitch) or G331 (rigid tapping).

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

Remedy: The thread pitch 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 pitch) the pitch parameter was not assigned to the axis that determines the velocity.
For longitudinal and face threads, the thread pitch 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.
Example:
N100 G33 Z400 K2 ; thread pitch 2mm, thread end Z=400 mm
N200 SPOS=0 ; position spindle in axis mode
N201 G90 G331 Z-50 K-2 ; tapping to Z=-50, counterclockwise
N202 G332 Z5 ; retraction, direction reversal automatic
N203 S500 M03 ; spindle again in spindle mode

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.

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 loaded only after a spindle 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.

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 given by the sign of the spindle lead:
- Positive thread pitch: Rotational direction as M03.
- Negative thread pitch: Rotational direction as M04 N2.

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: Incorrect programming: the spindle function can currently not be executed. This alarm occurs when the spindle (master 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.

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: One of the following spindle functions has been programmed, the position control requires:
SPCON,
SPOS, SPOSA,
COUPON,
G331/G332.
The position control requires at least one measuring system.
No measuring system has been configured in MD30200 $MA_NUM_ENCS of the programmed spindle.

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

Remedy: Please inform the authorized personnel/service department. Retrofit a measuring system.

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

16771 [Channel %1:] Block %3 following axis %2 overlaid movement not enabled

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

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 NC/PLC interface signal <Freigabe_Folgeachsueberlagerung/> (enable following axis overlay).

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 slave axis in a coupling. In the REF operation mode, the coupling is opened.
The alarm can be suppressed by means of MD11410 $MN_SUPPRESS_ALARM_MASK Bit29 = 1.

Reaction: Alarm display.

Remedy: The coupling will be closed again after having exited the REF operation mode.

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

16777 [Channel %1:] Block %2 coupling: following axis %3 for lead axis %4 not available

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Axis name, spindle number

Definitions: A coupling has been switched on in which the slave spindle/axis is currently not available. Possible causes:
- The spindle/axis is active in the other channel.
- The spindle/axis has been accessed by the PLC and has not yet been released.

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 results in a cyclic coupling, allowance being made for further couplings. This cyclic coupling cannot be uniquely computed.

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. Configure link in accordance with the MD or correct NC part program (MD21300 $MC_COUPLE_AXIS_1).

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

16786 [Channel %1:] Block %2 coupling to master spindle %3 already exists

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

Definitions: A coupling is to be switched on, in which the slave axis is already actively coupled with the other master axis. Only one master spindle is allowed for the synchronous spindle function. The already active master spindle is displayed as last alarm parameter.

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

Remedy: Prior to switching on the new coupling, separate the existing coupling. If several master spindels/axes are required, the ELG function will have to be used.

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

16800 [Channel %1:] Block %2 traverse instruction DC/CDC for axis %3 not allowed

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

Definitions: The keyword DC (Direct Coordinate) can only be used for rotary axes. This causes approach of the programmed absolute position along the shortest path.
Example:

N100 C=DC(315)

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

Remedy: Please inform the authorized personnel/service department.
Replace the keyword DC in the displayed NC block by specifying AC (Absolute Coordinate).
If the alarm display is the result of an error in the axis definition, the axis can be declared as a rotary axis by means of the axis-specific MD30300 $MA_IS_ROT_AX.
Corresponding machine data:
Modify MD30310: $MA_ROT_IS_MODULO
Modify MD30320: $MA_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

Definitions: The keyword ACP (Absolute Coordinate Positive) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

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 keyword ACP by specifying AC (Absolute Coordinate).
If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD30300 $MA_IS_ROT_AX and MD30310 $MA_ROT_IS_MODULO can be declared a rotary axis with modulo change.
Corresponding machine data:
Modify MD30320 $MA_DISPLAY_IS_MODULO

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

16820 [Channel %1:] Block %2 traverse instruction ACN for axis %3 not allowed

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

Definitions: The keyword ACN (Absolute Coordinate Negative) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.

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 keyword ACN by specifying AC (Absolute Coordinate).
If the alarm display is based on an incorrect axis definition, the axis with the axis-specific machine data MD30300: $MA_IS_ROT_AX and MD30310: $MA_ROT_IS_MODULO can be declared a rotary axis with modulo change.
Corresponding machine data:
MD30320: $MA_DISPLAY_IS_MODULO

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

16830 [Channel %1:] Block %2 incorrect position programmed for axis/spindle %3

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

Definitions: A position beyond the range of 0 - 359.999 has been programmed for a modulo axis.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Program position in the range 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
Definitions: The relevant action cannot be processed now. This can occur, for instance, during read-in of machine data.
Reaction: Alarm display.
Remedy: Wait until the procedure is terminated or abort with Reset and repeat the operation.
Program Continuation: Clear alarm with the Delete key or NC START.

16904 [Channel %1:] Program control: action %2<ALNX> not allowed in the current state

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The operation (program, JOG, block search, reference point, etc.) cannot be started or continued in the current status.
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
Definitions: Operation cannot be started or continued. A start is only accepted when an NCK function can be started.
Example: A start is accepted in JOG mode when, for example, the function generator is active or a JOG movement has first been stopped with the Stop key.
Reaction: Alarm reaction in Automatic mode.
Remedy: Check the program status and channel status.
Program Continuation: Clear alarm with the Delete key or NC START.

16906 [Channel %1:] Program control: action %2<ALNX> is aborted due to an alarm

Parameters: %1 = Channel number
%2 = Action number/action name
Definitions: The action was aborted due to an alarm.
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
Definitions: This action may only be performed in Stop state.
Reaction: Alarm display.
Remedy: Check the program status 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

Definitions: This action may only be performed in Reset state or at end of block.
In JOG mode, no axis that is traversed as geometry axis in the switched coordinate system, must be active as PLC or command axis (started through static synchronized action) on mode change. This means that axes like that must be in the state 'neutral axis' again.

Reaction: Alarm display.

Remedy: Check the program status and channel status.
Check in JOG mode whether the axes are PLC or command axes.

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

16909 [Channel %1:] Action %2<ALNX> not allowed in current mode

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

Definitions: A different operating mode must be activated 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: After overstoring is terminated, it is possible to change to another operating state again.

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

16912 [Channel %1:] Program control: action %2<ALNX> only possible in reset state

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

Definitions: This action can only be performed in Reset state.
Example: Program selection through 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

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 MDI mode!

Reaction: Alarm display.

Remedy: Either activate the Reset key to reset program processing, or activate the mode in which the program was being processed previously.

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

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

Definitions: Incorrect mode change, e.g.: Auto -> MDIREF.

Reaction: Alarm display.

Remedy: Check operation or selected mode.

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

16915 [Channel %1:] Action %2<ALNX> not allowed in the current block

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

Definitions: If traversing blocks are interrupted by asynchronous subroutines, then it must be possible for the interrupted program to continue (reorganization of block processing) after termination of the asynchronous subroutine.
The 2nd parameter describes which action wanted to interrupt block processing.

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

Definitions: Repositioning of block processing is presently not possible. A mode change cannot take place.
The 2nd parameter describes which action should be used to perform repositioning.

Reaction: Alarm display.

Remedy: Let the program continue to a repositioned NC block or modify part program.

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

16919 [Channel %1:] Action %2<ALNX> is not allowed due to a pending alarm

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

Definitions: This action cannot be performed due to an alarm, or the channel is in the fault condition.

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

Definitions: An identical action is still being processed.

Reaction: Alarm display.

Remedy: Wait until the previous procedure has been terminated 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

Definitions: Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines are activated. These asynchronous subroutines can be interrupted in the same manner as user programs. Unlimited nesting depth is not possible for asynchronous subroutines due to memory limitations.
Example: An interrupt interrupts the current program processing. Other interrupts with higher priorities interrupt processing of the previously activated asynchronous subroutines.
Possible actions are: DryRunOn/Off, DecodeSingleBlockOn, delete distance-to-go, interrupts .....

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

16923 [Channel %1:] Program control: action %2<ALNX> not allowed in the current state

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

Definitions: The current processing cannot be stopped since a preprocessing process is active.
This applies, for example, to the loading of machine data, and in block searches until the search target is found.

Reaction: Interface signals are set.
Alarm display.

Remedy: Cancel by pressing RESET!

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

16924 [Channel %1:] Caution: program test modifies tool management data

Parameters: %1 = Channel number

Definitions: Tool management data is changed during program testing. It is not possible to automatically rectify the data after termination of the program testing.
This error message prompts the user to make a backup copy of the data or to reimport the data after the operation is terminated.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.
Save tool data on 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
%3 = Action number/action name

Definitions: The action has been refused since a mode or sub-mode change (change to automatic mode, MDI, JOG, overstoring, digitizing, etc.) is taking place.
Example: This alarm message is output if the Start key is pressed during a mode or sub-mode change from, for example, automatic to MDI, before the NCK has confirmed selection of the mode.

Reaction: Alarm display.

Remedy: Repeat action.

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

16927 [Channel %1:] Action %2<ALNX> at active interrupt treatment not allowed

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

Definitions: This action may not be activated during interrupt processing (e.g. mode change).

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 number
%2 = Action number/action name

Definitions: A program interrupt has been activated in a non REORG capable block.
Examples of possible program interrupts in this case:
- Traversing to fixed stop
- Vdi channel delete distance-to-go
- Vdi axial delete distance-to-go
- Measuring
- Software limit
- Axis interchange
- Axis comes from tracking
- Servo disable
- Gear stage change at actual gear stage unequal to setpoint gear stage.

The block affected is a:
- collection block from block search (except for the last collection block)
- Block in overstore interrupt.

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 functions WAITMC, SETM, CLEARM and MSG must be packed in separate NC blocks due to the language definition. To avoid velocity drops, these blocks are attached to the next NC block internally in the NCK (for MSG only in path control mode, for WAITMC to the previous NC block). For this reason, there must always be an executable block (not a calculation block) between the NC blocks. An executable NC block always includes e.g. travel movements, a help function, Stopre, dwell time etc.

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.
Example:
N10 SETM.
N15 STOPRE ; insert executable NC block.
N20 CLEARM.

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

16931 [Channel %1:] Subprograms: action %2<ALNX> maximum nesting depth exceeded

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

Definitions: Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines (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: In the case of an approach block in a repositioning procedure do not interrupt repeatedly, instead wait until processing is completed.

Possible actions are: mode change, SlashOn/Off, overstoring.

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 zero offset or base frame) which is also written by the NC blocks in preparation.
In the event of a conflict, the value entered by the HMI is reset.
Parameter %2 specifies which data block is affected:
1: Active tool offset
2: Base frame
3: Active zero offset

Reaction: Alarm display.

Remedy: Check the inputs on the HMI 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 number
%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 the RESET key.

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

16934 [Channel %1:] Interrupt treatment: action %2<ALNX> not possible due to stop

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

Definitions: Reorg events are, e.g. abort subprogram, delete distance to go and interrupts, axis exchange, termination of follow-up mode. Two Reorg events overlap in this situation. The 2nd Reorg event coincides with the 1st block generated by the previous event. (e.g. an axis exchange is induced twice in rapid succession). Axis exchange leads to Reorg in the channels in which an axis is removed without preparation. This block must be stopped in the above sequence in order to prevent the interpolator buffer from overflowing. This can be achieved by pressing the Stop or StopAll key, configuring an alarm with INTERPRETERSTOP or by decode single block.

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

Remedy: The program must be aborted with Reset.

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

16936 [Channel %1:] Action %2<ALNX> not possible due to active dry run

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

Definitions: This action is not allowed as dry run feedrate is currently active.
Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when dry run feedrate is active.

Reaction: Alarm display.

Remedy: Abort program with the RESET key.

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

16937 [Channel %1:] Action %2<ALNX> not possible due to program test

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

Definitions: This action is not allowed as program test is currently active.
Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when program test is active.

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 number
%2 = Action number/action name

Definitions: Reorganization events are, among others, subprogram abort, delete distance-to- go and interrupts, axis exchange, exiting the correction 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: Program must be aborted with Reset and, if necessary, MD10192 $MN_GEAR_CHANGE_WAIT_TIME must be increased.

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

16939 [Channel %1:] Action %2<ALNX> rejected due to active gear change

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

Definitions: Reorganization events that are possible in Stop state, e.g mode change, are waiting for the end of the gear change. However, the maximum waiting period has elapsed.

Reaction: Interface signals are set.
Alarm display.

Remedy: Repeat action or increase MD10192 $MN_GEAR_CHANGE_WAIT_TIME.

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

16940 [Channel %1:] Action %2<ALNX> wait for gear change

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

Definitions: Reorganization events are waiting for the end of a gear change. The alarm is displayed during the waiting period.

Reaction: Alarm display.
Warning display.

Remedy: Alarm is suppressed by means of MD11411 $MN_ENABLE_ALARM_MASK bit 1 = 0.

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 number
%2 = Action number/action name

Definitions: The setting of the MD20108 $MC_PROG_EVENT_MASK forces an asynchronous subprogram to be triggered automatically on RESET or PowerOn. The implicitly triggered asynchronous subprograms are normally called "Event-triggered program call" or "Program event".
In the alarm situation, this asynchronous subprogram could not yet be activated; that is why the action (normally start of part program) must be rejected.
Reasons for the fact that the asynchronous subprogram could not be triggered:
1. The asynchronous subprogram does not exist (/_N_CMA_DIR/_N_PROG_EVENT_SPF)
2. The asynchronous subprogram is allowed to start in the referenced state only (see MD11602 $MN_ASUP_START_MASK)
3. READY is missing (because of alarm)

Reaction: Alarm display.

Remedy: Load program
Check MD11602 $MN_ASUP_START_MASK
Acknowledge alarm

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 number
%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 number
%2 = Action number/action name

Definitions: The currently executing action (e.g. dry run on/off, change skip levels, etc.) should be active immediately, but it can become active not earlier than at the end of the block, since a thread is currently being machined. The action is activated with a slight delay.
Example: Dry run is started in the middle of the thread, then traversing at high speed does not start before the next block.

Reaction: Alarm display.

Remedy: Alarm can be switched off via MD11410 $MN_SUPPRESS_ALARM_MASK bit17==1.

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

16950 [Channel %1:] Search run with hold block

Parameters: %1 = Channel number

Definitions: Informational alarm.
The search run was not performed on the interruption block, instead, it touches down shortly before that. This so-called "hold block" is generated by the part program command IPTRLOCK, or implicitly defined by MD22680 $MC_AUTO_IPTR_LOCK. This is to prevent you from performing a search run in critical program areas (e.g. gear hobbing). The alarm also displays that, instead of searching for the block that actually was interrupted before, another block is being searched for. This behavior is desired and the alarm serves only informational purposes.

Reaction: Alarm display.

Remedy: MD11410 $MN_SUPPRESS_ALARM_MASK, MD22680 $MC_AUTO_IPTR_LOCK and language command IPTRLOCK

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

16951 [Channel %1:] Search run in a protected program section.

Parameters: %1 = Channel number

Definitions: A part programmer can define protected part program sections with the language commands IPTRLOCK and IPTRUNLOCK. Every search run in these program sections will then be acknowledged with alarm 16951. In other words: When the alarm appears, the user has started a search run (Serupro type) and the search target lies in a protected area. A protected area can also be defined implicitly with the MD22680 $MC_AUTO_IPTR_LOCK.
Note:
The alarm can only be generated if the simulation has been completed during the search run. The alarm cannot be output immediately at the start of the search run.

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

Remedy: MD11410 $MN_SUPPRESS_ALARM_MASK, MD22680 $MC_AUTO_IPTR_LOCK and language command IPTRLOCK

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 number

Definitions: NCK is currently executing an ASUB in MDI mode. In this constellation, parts program command "Start" is not allowed for another channel. Attention: If an asup is started from JOG, the NCK can internally change to MDI, if the NCK was previously in MDI and not in RESET. Note: Without this alarm, the MDI 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 ASUB in AUTO or ->JOG in AUTO

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 number
%2 = Block number, label

Definitions: In a program area (stop delay area) that is bracketed with DELAYFSTON and DELAYFSTOF, a program command was used that causes a stop. No commands other than G4 are permissible that might cause a stop even though only shortly. A stop delay area can also be defined by MD11550 $MN_STOP_MODE_MASK.

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

Remedy: MD11550 $MN_STOP_MODE_MASK and language 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 number

Definitions: In a program area (stop delay area) that is bracketed by DELAYFSTON and DELAYFSTOF, an event has been detected that causes a stop. The stop is delayed and executed after DELAYFSTOF. A stop delay area can also be defined by MD11550 $MN_STOP_MODE_MASK.

Reaction: Interface signals are set.
Alarm display.

Remedy: MD11550 $MN_STOP_MODE_MASK and language command DELAYFSTON DELAYFSTOF

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

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 MD28020 $MC_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.
- Modify machine data
- Reduce the number of symbols (variables, subroutines, parameters)

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

17001 [Channel %1:] Block %2 no memory left for tool/magazine data

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

Definitions: The number of following tool/magazine data variables in the NC are given by machine data:
- Number of tools + number of grinding data blocks: MD18082 $MN_MM_NUM_TOOL
- Number of cutting edges: MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA
Tools, grinding data blocks, cutting edges can be used independently of the tool management.
The memory for the following data is available only if the corresponding bit has been set in MD18080 $MN_MM_TOOL_MANAGEMENT_MASK.
- Number of monitoring data blocks: MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA
- Number of magazines: MD18084 $MN_MM_NUM_MAGAZINE
- Number of magazine locations: MD18086 $MN_MM_NUM_MAGAZINE_LOCATION
The following variable is determined by software configuration: Number of magazine spacing data blocks: P2 permits 32 such spacing data blocks.
Definition:
- 'Grinding data blocks': Grinding data can be defined for a tool from type 400 to 499. Such a data block occupies as much additional memory as that provided for a cutting edge.
- 'Monitoring data blocks': Each cutting edge of a tool can be supplemented by monitoring data.
- If the alarm occurs while writing from one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR, check whether machine data MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC / MD18076$MN_MM_NUM_LOCS_WITH_DISTANCE have been correctly set.
MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC defines the number of different Index1 statements that may be made for an Index2 value.
MD18076 $MN_MM_NUM_LOCS_WITH_DISTANCE defines the number of different buffer storage locations that may be named in Index2.

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

Remedy: Please inform the authorized personnel/service department.
- Modify machine data
- Modify NC program, i.e. reduce number of rejected variable.

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

17010 [Channel %1:] Block %2 no memory left

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

Definitions: When executing/reading files from the active working memory, it was found that there is not enough memory space (e.g. for large multidimensional arrays or when creating tool offset memory).

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

Alarm display.

Remedy: Please inform the authorized personnel/service department. Make arrays smaller or make more memory space available for memory management of subroutine calls, tool offsets and user variables (machine data MM_...).
See /FB/, S7 Memory Configuration

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

17020 [Channel %1:] Block %2 illegal array index 1

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

Definitions: General:
A read or write access has been programmed to an array variable with an invalid 1st array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
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.
The following applies specifically: If the alarm occurs while writing from one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR,
check whether MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC has been set correctly.
MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC defines the number of different Index1 statements that may be made for an Index2 value.

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. If an SPL is used in Safety Integrated, the field index via optional data may be subject to additional restrictions.

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: General:
A read or write access has been programmed to an array variable with an invalid 2nd array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
PROFIBUS I/O:
An attempt was made to read/write data outside the slot / I/O area limits of the stated slot / I/O area.
The following applies specifically: If the alarm occurs while writing one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR,
check whether MD18076 $MN_MM_NUM_LOCS_WITH_DISTANCE has been set correctly.
$MN_MM_NUM_LOCS_WITH_DISTANCE defines the number of different buffer storage locations that may be named in Index2.

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.
Example:
Writing of an axial machine data

$MA_... [X]= ... ; but geometry axis X cannot be imaged on a machine axis because of a transformation!

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

Remedy: Deselect transformation before writing into the axial data (keyword: TRAFOOF) or use the machine axis names as axis index.

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

17050 [Channel %1:] Block %2 illegal value

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

Definitions: A value has been programmed that exceeds the value range or the limit value of a variable or a machine data.
Example: - in a string variable (e.g. GUD or LUD) a string shall be written that exceeds the string length agreed upon in the variable definition.
- if an invalid value is to be written in a tool or magazine management variable (e.g. invalid cutting edge number in $TC_DPCE[x,y] or invalid magazine location number in $TC_MDP2[x,y]).
- an invalid value is to be written in $P_USEKT or $A_DPB_OUT[x,y].
- an invalid value is to be written in a machine data (e.g. MD10010 $MN_ASSIGN_CHAN_TO_MODE_GROUP[0] = 0).
- on accessing an individual frame element, a frame component other than TRANS, ROT, SCALE or MIRROR was addressed or the function CSCALE has been given a negative scale factor.

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

Remedy: Address frame components only with the keywords provided; program the scale factor between the limits of 0.000 01 to 999.999 99.

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

17060 [Channel %1:] Block %2 requested data area too large

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

Definitions: The maximum memory space of 8 KB available for a symbol has been exceeded.

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 was made to write into a write-protected variable (e.g. a system variable). Safety Integrated: Safety system variables can only be written into via the safety SPL program.

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

Remedy: Please inform the authorized personnel/service department. Modify part program.

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

17080 [Channel %1:] Block %2 %3 value below lower limit

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

Definitions: An attempt was made to write into a machine data with a value 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 machine data and assign a value within these limits.

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

17090 [Channel %1:] Block %2 %3 value exceeds upper limit

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

Definitions: An attempt was made to write into a machine data with a value greater 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 machine data and assign a value within these limits.

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

17095 [Channel %1:] Block %2 invalid value

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

Definitions: An attempt was made to write an invalid value, e.g. zero, into a machine data.

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 = Input number

Definitions: Either an attempt was made to read a digital input n via the system variable $A_IN[n] and this input has not been activated via NCK MD10350 $MN_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

Definitions: An attempt was made to read or set a digital NCK output (connector X 121) via the system variable $A_OUT [n] with the index [n] greater than the specified upper limit in MD10360 $MN_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 MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS.

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

17120 [Channel %1:] Block %2 analog input no. %3 not activated

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

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 MD10300 $MN_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 MD10310 $MN_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 = Quantity

Definitions: No more than the specified number of outputs may be programmed in an NC block.
The quantity of hardware outputs is defined in the MDs:
MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS and
MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS

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

Remedy: Program fewer digital/analog outputs in a block. The specified maximum number applies in each case separately for analog or digital outputs. If necessary, program two NC blocks.

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

17160 [Channel %1:] Block %2 no tool selected

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

Definitions: An attempt has been made to access the current tool offset data via the system variables:
$P_AD [n]: Contents of the parameter (n: 1 - 25)
$P_TOOL: Active D number (tool edge number)
$P_TOOLL [n]: Active tool length (n: 1- 3)
$P_TOOLR: Active tool radius
although no tool had been selected previously.

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 program before using the system variables.
Example:
N100 G.. ... T5 D1 ... LF
With the channel-specific machine data:
Modify MD22550 $MC_TOOL_CHANGE_MODE
New tool offset for M function
Modify MD22560 $MC_TOOL_CHANGE_M_CODE
M function with tool change
It is established whether a tool offset is activated in the block with the T word or whether the new offset values are allowed for only when the M word for tool change occurs.

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

17170 [Channel %1:] Block %2 number of symbols too large

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

Definitions: The predefined symbols could not be read in during power-up.

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

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 that is not defined and therefore is not available.

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

Remedy: Check tool call in the NC parts program:
- Correct tool correction number D programmed? If no D number is specified with the tool change command, then the D number set by MD20270 $MC_CUTTING_EDGE_DEFAULT will be active automatically. It is D1 by default.
- Tool parameters (tool type, length,...) defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through a tool data file in NCK.
Description of the system variables $TC_DPx[t, d] as included in a tool data file.
x ... Correction parameter number P
t ... Associated tool number T
d ... Tool correction number D

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

17181 [Channel %1:] Block %2 T no.= %3, D no.= %4 not existing

Parameters: %1 = Channel number
%2 = Block number, label
%3 = T number
%4 = D number

Definitions: A programmed D number was not recognized by the NC. By default, the D number refers to the specified T number. If the flat D number function is active, T= 1 is output.

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.

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 (except for H=0) must be assigned in a TO unit only once. The indicated edge already has the H number. If the H number shall be assigned more than once, MD10890 $MN_EXTERN_TOOLPROG_MODE, bit 3 must be set = 1.

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

Remedy: - Change program:
- Select different 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 TO unit of channel %1 is not unique. The specified T numbers %3 and %4 each have an offset with number %2. If tool management is active: The specified T numbers belong to tool groups with different names.

Reaction: Interface signals are set.
Alarm display.

Remedy: 1. Ensure that the D numbers within the TO unit are unique.
2. If unique numbering is not necessary for subsequent operations, do not use the command.

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

17190 [Channel %1:] Block %2 illegal T number %3

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

Definitions: In the displayed block, access is made to a tool that is not defined and therefore not available. The tool has been named by its T number, its name or its name and duplo number.

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 through the operator panel or through the V.24 interface.
Description of the system variables $P_DP x [n, m]
n ... Associated tool number T
m ... Tool edge number D
x ... Parameter number P

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

17191 [Channel %1:] Block %2 T= %3 not existing, program %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = T number or T identifier
%4 = Program name

Definitions: A tool identifier which the NCK does not recognize was programmed.

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 NCK (via HMI with overstore) and continue the program.
If the program pointer is at an NC block which does not contain the specified T identifier: The error occurred at an earlier point in the program where the T command appeared, but the alarm was not output until the change command was detected.
If the program is incorrect - T5 programmed instead of T55 - the current block can be corrected with a correction block; i.e. if only M06 is entered, you can correct the block with T55 M06. The incorrect T5 line remains in the program until it is terminated by a RESET or end of program.
In complex program structures with indirect programming, it may not be possible to correct the program. In this case, you can only intervene locally with an overstore block - with T55 in the example. If the data block is missing, create one. You can do this by downloading the data block of the tool with all the defined D numbers onto the NCK (via HMI with overstore), program the T command with overstore, and continue the program.

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

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 command correctly programmed?
- Does the status of the tool prevent access?

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

17200 [Channel %1:] Block %2: Data of tool %3 cannot be deleted.

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

Definitions: An attempt has been made to delete from the part program the tool data for a tool currently being processed. Tool data for tools involved in the current machining operation may not be deleted. This applies both for the tool preselected with T or that has been changed in place of another, and also for tools for which the constant grinding wheel peripheral speed or tool monitoring is active.

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

Remedy: Check access to tool offset memory by means of $TC_DP1[t,d] = 0 or deselect tool.

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

17210 [Channel %1:] Block %2 access to variable not possible

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

Definitions: The variable cannot be written/read directly from the part program. It is allowed only in motion synchronous actions.
Example for variable:
$P_ACTID (which planes are active)
$AA_DTEPB (axial distance-to-go for reciprocating infeed)
$A_IN (test input)
Safety Integrated: Safety PLC system variables can only be read during the safety SPL startup phase.

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.

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: Modify NC program: Assign the value of the machine data or of the system variable to a program-local variable and transfer this as parameter.

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

17500 [Channel %1:] Block %2 axis %3 is not an indexing axis

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

Definitions: An indexing axis position has been programmed for an axis with the keywords CIC, CAC or CDC that has not been defined as indexing axis in the machine data.

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:
Modify MD30500: $MA_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. 2 tables are possible (input value 1 or 2).
Modify MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1
Modify MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2 (Number of positions for 1st/2nd indexing axis)
Standard value: 0 Maximum value: 60
Modify MD10910 $MN_INDEX_AX_POS_TAB_1 [n]

Modify MD10930 $MN_INDEX_AX_POS_TAB_2 [n]
(Positions of the 1st indexing axis) The absolute axis positions are entered. (The list length is defined via MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1).

Program Continuation: Clear alarm with NC START or RESET key and continue the 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 setting the actual value memory (PRESET) is not possible for this axis.
Example:
Machine axis A should be set to the new actual value A 100 at the absolute position A 300.
:
N100 G90 G00 A=300
N101 PRESETON A=100

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 the keyword TRAFOOF.

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

17610 [Channel %1:] Block %2 axis %3 involved in the transformation, action cannot be carried out

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

Definitions: The axis is involved in the active transformation. It can therefore not execute the demanded action, traversing as positioning axis, enable for axis exchange.

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

Remedy: Deselect the transformation with TRAFOOF ahead of time or remove the action from the part program block

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

17620 [Channel %1:] Block %2 approaching fixed point for transformed axis %3 not possible

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

Definitions: In the displayed block, an axis is programmed for the fixed point approach (G75) that is involved in the active transformation. Fixed point approach is not performed with this axis!

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

Remedy: Remove G75 instruction from the part program block or previously deselect transformation with TRAFOOF.

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

17630 [Channel %1:] Block %2 referencing for transformed axis %3 not possible

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

Definitions: In the displayed block, an axis is programmed for reference point approach (G74) that is involved in the active transformation. Reference point approach is not performed with this axis!

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 previously deselect the transformation with TRAFOOF.

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

17640 [Channel %1:] Block %2 spindle operation for transformed axis %3 not possible

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

Definitions: The axis programmed for the spindle operation is involved in the current transformation as geometry axis. This is not allowed.

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.

17650 [Channel %1:] Block %2 machine axis %3 not programmable

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

Definitions: The machine axis cannot be used in an active transformation. You may be able to program the function in a different coordinate system. For example, it may be possible to specify the retraction position in the basic coordinate system or the workpiece coordinate system. The axis identifier is used to select the coordinate system.

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 keyword FP=n is not permissible. Two (2) absolute axis positions can be directly defined as fixed points by the axis-specific machine data MD30600 $MA_FIX_POINT_POS[n].
Or, if position numbers 3 and/or 4 are to be used, then machine data MD30610 $MA_NUM_FIX_POINT_POS must be set accordingly.

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

Remedy: Program keyword FP with machine fixed points 1 or 2.
Example:
Approach fixed point 2 with machine axes X1 and Z2.
N100 G75 FP=2 X1=0 Z2=0
Or modify MD30610 $MA_NUM_FIX_POINT_POS and, if necessary, MD30600 $MA_FIX_POINT_POS[].

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 (MD10000 $MN_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

18100 [Channel %1:] Block %2 invalid value assigned to FXS[]

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

Definitions: The following values are valid at the present time:
0: "Deselect traverse against fixed stop"
1: "Select traverse against fixed stop" valid.

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.

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 shift to settable frames or the basic frame is not possible since MD18600 $MN_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 MD18600 $MN_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 NCU 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.

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

18313 [Channel %1:] Block %2 frame: illegal switchover of geometry axes

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

Definitions: It is not allowed to change the geometry axis assignment because the current frame contains rotations.

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 MD10602 $MN_FRAME_GEOAX_CHANGE_MODE.

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

18314 [Channel %1:] Block %2 frame: type conflict

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

Definitions: It is not possible to chain global frames and channel-specific frames. The alarm occurs if a global frame is programmed with a channel axis name and no machine axis on this NCU is assigned to the channel axis. Channel-specific frames cannot be programmed with machine axis names if there is no corresponding channel axis on this NCU.

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 are possible (see parameter 3):
1. Invalid 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.

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 MD34030 $MA_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 MD34030 $MA_REFP_MAX_CAM_DIST is too small.
Determine the maximum possible distance from the beginning of reference motion up to the reduction cam and compare with the value in MD34030 $MA_REFP_MAX_CAM_DIST, increase the value in the MD if necessary.
2. The cam signal is not received by the PLC input module.
Operate the reference point switch manually and check the input signal on the NC/PLC interface (route: Switch! Connector! Cable! PLC input! User program).
3. The reference point switch is not operated by the cam.
Check the vertical distance between reduction cam and activating switch.

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

20001 [Channel %1:] Axis %2 no cam signal present

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

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: Please inform the authorized personnel/service department. Check whether the deceleration path after the approach velocity is greater than the distance to the reference point cam - in which case the axis cannot stop until it is beyond the cam. Use a longer cam or reduce the approach velocity in MD34020 $MA_REFP_VELO_SEARCH_CAM.
When the axis has stopped at the cam, it must be checked whether the signal V380x1000.7 (Deceleration reference point approach) is still available at the interface to the NCK.
- 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 hardware zero mark of the incremental position encoder or the substitute zero mark of the absolute position encoder is not within a defined section.
Phase 2 of the reference point approach ends when the zero mark of the encoder has been detected after the rising/falling edge of the NC/PLC interface signal V380x1000.7 (Deceleration reference point approach) has given the trigger start. The maximum distance between the trigger start and the zero mark that follows is defined in the MD34060 $MA_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.

Remedy: Please inform the authorized personnel/service department.
Check the cam adjustment and make sure that the distance is sufficient between the end of the cam and the zero marker signal that follows. The path must be greater than the axis can cover in the PLC cycle time.
Increase the MD34060 $MA_REFP_MAX_MARKER_DIST, but do not select a value greater than the distance between the 2 zero markers. This might result in the monitor being switched off.

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

20003 [Channel %1:] Axis %2 measuring system error

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

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 value entered in MD34300 $MA_ENC_REFP_MARKER_DIST. The control does not issue the alarm until it has again detected a distance that is too long after having made a 2nd attempt in reverse direction with half the traversing velocity.

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 MD34060 $MA_REFP_MAX_MARKER_DIST.
Check the reference track of the scale including the electronics for the evaluation.

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

20004 [Channel %1:] Axis %2 reference mark missing

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

Definitions: In the distance-coded length measurement system, 2 reference marks were not found within the defined searching distance (axis-specific MD34060 $MA_REFP_MAX_MARKER_DIST).
A reduction cam is not required for distance-coded scales (but an existing cam will be evaluated). The conventional direction key determines the direction of search.
The searching distance MD34060 $MA_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 point markers (reference point marker interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in the MD34060 $MA_REFP_MAX_MARKER_DIST.
Check the reference point track of the scale including the electronics for the evaluation.

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

20005 [Channel %1:] Axis %2 reference point approach aborted

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

Definitions: Referencing could not be completed for all stated axes (e.g., abort caused by missing servo enable, measuring system switchover, release of direction key, etc.).
In distance-coded measuring systems, the alarm will also be displayed if the value 1 has been set in MD34000 $MA_REFP_CAM_IS_ACTIV (reference cams) and one of the conditions stated in the remedy has been fulfilled.

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 possible reasons for termination:
- Servo enable missing: NC/PLC interface signal V380x 0002.1 (Servo enable)
- Measuring system switchover: NC/PLC interface signal V380x 0001.5 / 1.6 (Position measuring system 1/2)
- Traversing key + or - missing: NC/PLC interface signal V380x 0004.7 / 4.6 (Traversing keys plus/minus)
- Feed override = 0
- The feed disable is active
- Exact stop not reached within MD36020 $MA_POSITIONING_TIME.

The axis-specific MD34110 $MA_REFP_CYCLE_NR determines which axes are involved in the channel-specific referencing.
ValueMeaning
-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. Phase 1 is therefore considered as being already 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 reached at the 2nd attempt either, referencing will be stopped and the alarm displayed.
Approach velocity: MD34040 $MA_REFP_VELO_SEARCH_MARKER
Velocity tolerance: MD35150 $MA_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 MD34040 $MA_REFP_VELO_SEARCH_MARKER and/or increase the MD for the velocity tolerance MD35150 $MA_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 MD34200 $MA_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 reference mode MD34200 $MA_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: When setting MD34200 $MA_ENC_REFP_MODE = 6 the 2nd encoder must first be referenced.

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

Remedy: Modify referencing mode MD34200 $MA_ENC_REFP_MODE or reference 2nd encoder.

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

20050 [Channel %1:] Axis %2 handwheel mode active

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

Definitions: The axes cannot be traversed in JOG mode using the traversing keys because traversing is still taking place via the handwheel.

Reaction: Alarm display.

Remedy: Decide whether the axis is to be traversed by means of the direction keys or the handwheel. End handwheel travel and delete the axial distance-to-go if necessary (NC/PLC interface signal V380x 0002.2 (Delete distance-to-go/Spindle reset)).

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 jog 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 using the direction keys on the machine control panel. However, this is not possible because:
1. It is already traversing as a geometry axis (through the channel-specific interface V3200 1000.7 / 0.6 (Traversing keys -/+) or V3200 1004.7 / 4.6 (Traversing keys -/+) or V3200 1008.7 / 8.6 (Traversing keys -/+)) or
2. It is already traversing as a machine axis (through the axis-specific interface V380x 0004.7 / 4.6 (Traversing keys plus/minus)) or

3. A frame is valid for a rotated coordinate system, and another geometry axis involved in this is already traversing in JOG mode by means of the direction keys.

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.
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
MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1
MD10910 $MN_INDEX_AX_POS_TAB_1
MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2
MD10930 $MN_INDEX_AX_POS_TAB_2
or set the working area limits or the software limit switches to other values.

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

20055 [Channel %1:] Master spindle not present in JOG mode

Parameters: %1 = Channel number

Definitions: The displayed axis is to be traversed as machine axis in JOG mode with revolutional feed, but no master spindle has been defined from which the actual speed could have been derived.

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 MD20090 $MC_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 $MA_JOG_REV_VELO or in the case of rapid traverse overlay MD32040 $MA_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 spindle/axis the feed is to be derived from 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 the part program or
- Specify the correct feed for PLC axes at the VDI interface,
- Specify feed for oscillating axes in the SD43740 $SA_OSCILL_VELO.

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

20058 [Channel %1:] Axis %2 revolutional feedrate: illegal feed source

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

Definitions: An axis/spindle is to be traversed at revolutional feedrate. The reference axis/spindle defined in SD 43300 $SA_ASSIGN_FEED_PER_REV_SOURCE refers to itself. The coupling caused cannot be executed.

Reaction: Alarm display.

Remedy: The reference axis/spindle must be modified accordingly in SD 43300.

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

20060 [Channel %1:] Axis %2 cannot be traversed as geometry axis

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

Definitions: The axis is currently not in "Geometry axis" state. Therefore, it cannot be traversed in JOG mode as geometry axis.
If the abbreviation WCS (workpiece coordinate system) is displayed in the "Position" screen, then only the geometry axes can be traversed by means of the direction keys! (MCS ... Machine coordinate system; all machine axes can now be traversed by using the direction keys on the machine control panel).

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.

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 an axis can take place in JOG mode through 2 different interfaces.

1. As a geometry axis: via the channel-specific interface V3200 1000.7 / 0.6 (Traversing keys -/+)
2. As a machine axis: via the axis-specific interface V380x 0004.7 / 4.6 (Traversing keys plus/minus)
With the standard machine control panel, it is not possible to operate an axis as a machine axis and as a 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.

20065 [Channel %1:] Master spindle not defined for geometry axes in JOG mode

Parameters: %1 = Channel number

Definitions: The displayed axis is to be traversed as geometry axis in JOG mode with rotary feed, but no master spindle has been defined from which the actual speed could be derived.

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

Remedy: If the revolutional feedrate is to be active in JOG mode too, then a master spindle must be declared in the channel-specific machine data MD20090 $MC_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 feedrate, the control takes the value assigned in the axis-specific machine data MD32050 $MA_JOG_REV_VELO or in the case of rapid traverse override MD32040 $MA_JOG_REV_VELO_RAPID).
The revolutional feedrate in JOG mode is deactivated by changing the G function from G95 to G94.

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

20070 [Channel %1:] Axis %2 software limit switch %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"

Definitions: The axis is traversed by the PLC as a concurrent positioning axis and the corresponding software limit switch is violated for the axis. No traversing.
With an additional message to alarm 20140, the axis is traversed as a command axis.

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. Retract axis via JOG.

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" and the corresponding working area limitation active 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.

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.

20080 [Channel %1:] Axis %2 no handwheel assigned for override

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

Definitions: No handwheel has been assigned for this specified axis after handwheel overlay has been started in automatic mode. If the axis identifier is missing in the alarm with active velocity overlay FD > 0, then the 1st geometry axis has not been defined in the NC channel. In this case the block is executed without handwheel control.

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.

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 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: 1. The "Traverse against fixed stop" function has been programmed with FXS[AX]=1 but the axis does not (yet) support this. Check MD37000 $MA_FIXED_STOP_MODE. This function is not available for gantry axes and simulated axes.
2. On selection, no movement was programmed for axis AX. AX is a machine axis identifier.
3. It is always necessary to program a traversing movement in the selection block for the axis/spindle for which the "Traverse against fixed stop" function is activated.
The alarm can be reprogrammed in the MD11412 $MN_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.
- Check the axis type.
- Check MD37000 $MA_FIXED_STOP_MODE.
- Is a machine axis movement missing in the approach block?

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

20091 Axis %1 has not reached fixed stop

Parameters: %1 = Axis name, spindle number

Definitions: On attempting to traverse against a fixed stop, the programmed end position has been reached or the traversing movement has been aborted. The alarm can be concealed by means of the MD37050 $MA_FIXED_STOP_ALARM_MASK.
The alarm can be reprogrammed in the MD11412 $MN_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: Correct the part program and the settings:
- Has the traversing block been aborted?
- If the axis position does not correspond to the programmed end position, then correct the end position.
- If the programmed end position is in the part, the triggering criterion must be checked.
- Has the contour deviation leading to triggering been dimensioned too large? Has the torque limit been set too high?

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

20092 Axis %1 travel to fixed stop still active

Parameters: %1 = Axis name, spindle number

Definitions: An attempt has been made to move an axis while it is in fixed stop or while the deselection function has not yet been completed.
The alarm can be reprogrammed in the MD11412 $MN_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.
Check the following:
- Has the axis at the fixed stop also been moved by a traversing movement of geometry axes?
- Is a selection carried out even though the axis is stationary at the stop?
- Has the deselection process been interrupted by a RESET?
- Has the PLC switched the acknowledgement signals?

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

20093 Axis %1 standstill monitoring at fixed-stop end point has been triggered

Parameters: %1 = Axis name, spindle number

Definitions: The position of the axis has been beyond the zero speed window ever since selection has been completed.
The alarm can be reprogrammed in the MD11412 $MN_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.

- Check the mechanical components, e.g. has the stop broken away? Has the part to be clamped given way?
- Position window for zero speed control too small (MD37020 $MA_FIXED_STOP_WINDOW_DEF) (SD43520 $SA_FIXED_STOP_WINDOW). Default is 1 mm in each case.

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

20094 Axis %1 function has been aborted

Parameters: %1 = Axis name, spindle number

Definitions: The function has been aborted. The possible reasons for this are:
- Because a pulse disable has occurred, the torque can no longer be provided.
- The PLC has removed the acknowledgments.

The alarm can be reprogrammed in the MD11412 $MN_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: Check whether
- there is a pulse disable from the infeed/regenerative-feedback unit or from the PLC?
- the acknowledgement bits have been deleted by the PLC even though NCK has not requested deselection?

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

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

21550 [Channel %1:] Axis %2 Travel from hardware limit switch not possible. Reason: %3

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

Definitions: It has been tried to retract a following axis of an axis coupling or an output axis of a transformation through the master axis or input axis of a transformation. This is not permitted in the current situation.
Possible reasons:
1 No permissible direction of retraction
2 Coupling not synchronous
3 Retraction not permitted for the active coupling
4 Reserved
5 Retraction not permitted for the active transformation

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

Remedy: Remedy for error cause:
1 Define another travel direction
2 Deactivate the coupling and travel the axis/axes separately
3 Deactivate the coupling and travel the axis/axes separately

4 Reserved
5 Deactivate the transformation and travel the axis/axes separately

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

21610 [Channel %1:] Axis %2 encoder %3 frequency exceeded

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = String (encoder number)

Definitions: The maximum permissible frequency of the currently active encoder (axis-specific interface signal V380x 0001.5 / 1.6 (position measuring system 1/2)) in the axis-specific MD36300 $MA_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 reprogrammed in MD11412 $MN_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: Check MD36300 $MA_ENC_FREQ_LIMIT [n] and NC/PLC interface signal V380x 0001.5 / 1.6 (position measuring system 1/2).

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

21612 [Channel %1:] Axis %2: enable reset, cause %3

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Cause of the alarm

Definitions: Causes of alarm:
0: The cause of the alarm cannot be preceisely determined.
1: The interface signal V380x 0002.1 (Servo enable) is missing
2: The interface signal V380x 4001.7 (Pulse enable) is missing
3: Drive signal V390x 4001.7 (Impulses enabled) is not set
4: Drive signal V390x 4001.5 (Drive ready) is not set
One of the motion-enabling signals (e.g. "Servo enable", "Pulse enable", parking/encoder selection (only for axes) or drive-specific enables has been reset for the displayed axis. The alarm can be reported with positioning axes, spindles and for axes from the geometry grouping.
The axes entered in the channel-specific MD array MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB are regarded as axes belonging to the geometry grouping.
Servo enable must exist for all available geometry axes, regardless of whether or not they are currently in motion.
Occurs in connection with SAFETY function: If a test stop is performed with linked axes, the alarm is issued if a motion command from the ELG grouping is pending during the test stop of the slave axis.

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: Please inform the authorized personnel/service department.
Check the interface signals V380x 0002.1 (Servo enable), V380x 4001.7 (Pulse enable), check the drive signals V390x 4001.7 (Pulses enabled), V390x 4001.5 (Drive ready) for example with the PLC status display in the DIAGNOSTICS operating area. Check the encoder selection (for axes) as well as other signals enabling motion (such as SIMODRIVE 611D terminal 663 etc.) according to the drive type used.
When the terminal enables of the drive have failed, trace back the wiring or hardware function (for example relay function) or proceed as stated in the relevant drive documentation.
With SAFETY: With active actual-value linkage, output of the error message on the slave axis can be prevented by increasing MD36060 $MA_STANDSTILL_VELO_TOL (default value is 5 mm).

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

21613 Axis %1 measuring system changing

Parameters: %1 = Axis name, spindle number

Definitions: The measuring system for this axis is changing.

Reaction: Alarm display.

Remedy: -

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

21614 [Channel %1:] Axis %2 hardware limit switch %3

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

Definitions: The signal V380x 1000.1 und .0 (Hardware limit switch plus/minus) has been set at the NC/PLC interface.

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

Remedy: Please inform the authorized personnel/service department.
1. With axes that have already been referenced, the software limit switch 1 or 2 should respond before the hardware limit switch is reached. Check MD36110 $MA_POS_LIMIT_PLUS, MD36100 $MA_POS_LIMIT_MINUS, MD36130 $MA_POS_LIMIT_PLUS2 and MD36120 $MA_POS_LIMIT_MINUS2 and the NC/PLC interface signal for the selection V380x 1000.3 / 1000.2 (1st/2nd software limit switch plus/minus) and correct, if necessary (PLC user program).
2. If the axis has not yet been referenced, it is possible to depart from the hardware limit switch in the opposite direction in JOG mode.
3. Check the PLC user program and the connection from the switch to the PLC input module, provided the axis has not reached the hardware limit switch at all.

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

21615 [Channel %1:] Axis %2 taken from traverse mode to follow-up mode

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

Definitions: This axis has been taken from traverse mode and put into "Follow-up" mode, for instance because the pulse enable for the drive has been reset.

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

21617 [Channel %1:] Block %2 transformation does not allow to traverse the pole

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

Definitions: The preset curve passes through the pole or a forbidden area of the transformation.

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

Remedy: Modify the part program (if the 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).

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. Transformation-dependent error causes can be in:
TRANSMIT: A (circular) area exists around the pole, where positioning is not possible. The area is caused by the fact that the tool reference point cannot be traversed as far as into the pole. The area is defined by:
- the machine data (MD249.. $MC_TRANSMIT_BASE_TOOL_...)
- the active tool length compensation (see $TC_DP..).
Whether the tool length compensation is included in the calculation depends on the working plane selected (see G17,..). The machine stops at the edge of the area where positioning is not possible.

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

Remedy: Modify part program. Change the incorrectly specified tool length compensation.
Note: RESET alone is not enough if transformation also remains active during RESET.

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

21700 [Channel %1:] Block %3 axis %2 touch probe already deflected, edge polarity not possible

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

Definitions: The probe programmed under the keyword MEAS or MEAW is already deflected and has switched. For a further measuring operation, the probe signal must first be canceled (quiescent state of the probe).
The axis display is of no significance at the present time but an axis-specific evaluation has been planned for later stages of development.

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

Remedy: Verify the starting position of the measuring operation or check the probe signals in the PLC interface V2700 0001.0 / .1 (Probe actuated key 1/key 2). Are the cables and connectors in good order?

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 measurement mode
- Invalid probe
- Invalid encoder
- Invalid number of measurement signal edges
- Identical measurement signal edges are only programmable in mode 2
- Invalid FIFO number
- Mismatch between the number of FIFOs programmed and the number of probes used in the measurement task.

Further causes:
A measurement task is already active (e.g. from a synchronized action).

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

Remedy: Correct the measurement tasks.

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

21702 [Channel %1:] Block %3 axis %2 measurement aborted

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

Definitions: The measurement block has ended (the programmed end position of the axis has been reached) but the activated touch probe has not yet responded.
Measurement level 2 (MEAWA, MEASA, MEAC)
Measured values cannot be converted to the workpiece coordinate system. The measured values of the GEO axes programmed in the measurement task are only available in the machine coordinate system.
Causes:
Not all GEO axes were programmed in the measurement task. At least one measured value is therefore missing for conversion back into the workpiece coordinate system.
Further causes:
The measurement tasks programmed for all GEO axis are not identical.

Reaction: Alarm display.

Remedy: Verify the traversing movement in the measurements block.
- Is it necessary in all cases for the activated probe to have switched up to the specified axis position?
- Are the probe, cable, cable distributor, terminal connections in good order?

Either program all GEO axes explicitly or program the traversing movement with the POS[axis] command.

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

21703 [Channel %1:] Block %3 axis %2 touch probe not deflected, illegal edge polarity

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

Definitions: The selected probe is not (!) deflected and therefore cannot record any measured value from the deflected to the non-deflected state.
Measurement level 2 (MEAWA, MEASA, MEAC)
The degree of deflection of the probe at the start of the measurement task is identical to the first programmed measurement signal edge. The test is only performed in mode 2.

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 positioning for measuring
- Check program

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

21740 Output value at analog output no. %1 has been limited

Parameters: %1 = No. of output

Definitions: The value range of the analog output n is limited by MD10330 $MN_FASTIO_ANA_OUTPUT_WEIGHT[n].

Reaction: Alarm display.

Remedy: With $A_OUTA[..] = x no greater values can be programmed than permitted in the respective machine data.

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

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 has exceeded the maximum permissible amount. This alarm can occur in conjunction with motion synchronous actions: The maximum number of auxiliary functions must not be exceeded in motion block and motion synchronous actions.

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 setpoint

Definitions: This alarm is activated via MD27880 $MC_PART_COUNTER, bit 1: The number of counted workpieces ($AC_ACTUAL_PARTS or $AC_SPECIAL_PARTS) is equal or already greater than the programmed value for the number of required workpieces ($AC_REQUIRED_PARTS). At the same time, the channel VDI signal "Workpiece setpoint reached" is output. The value for the number of counted workpieces ($AC_ACTUAL_PARTS) is reset, while the value of $AC_SPECIAL_PARTS is retained.
Note:
The setpoint/actual comparisons of the workpieces are only made after an NC start
under the condition that $AC_REQUIRED_PARTS > 0. If $AC_REQUIRED_PARTS
has a negative value, all workpiece counts activated through MD27880 $MC_PART_COUNTER are frozen at the values they have reached,
and the nominal/actual comparison is discontinued.

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

Remedy: No program interrupt. Delete alarm display.

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

22000 [Channel %1:] Block %3 spindle %2 gear stage change not possible

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

Definitions: A gear stage change for the spindle will not be possible, if:
- thread cutting (G33, G34, G35) is active
- the spindle is active as master or slave spindle in a coupling
- the spindle is being positioned

Reaction: Interpreter stop

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

Remedy: The gear stage is to be set prior to the corresponding machining step.
If it is necessary, however, to change the gear stage within one of the above mentioned functions, this function must be switched off for the time of the gear stage change. Thread cutting is deselected with G1; synchronous spindle coupling is switched off with COUPOF; the spindle positioning operation is exited with M3, M4 or M5.

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 required gear stage called for by the NC. Note: Wherever possible, the requested gear stage should always be engaged.

Reaction: Alarm display.

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

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

22011 [Channel %1:] Block %3 spindle %2 change to programmed gear stage not possible

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

Definitions: With the 'DryRun', 'ProgramTest' and 'SearchRunByProgTest' functions deselected, it is not possible in the Repos module to carry out a gear stage change to a previously programmed gear stage. This is the case, if the spindle is in the deselection block not active in speed control mode, as a slave axis or in a transformation. Execution of a gear stage change is avoided if the above mentioned functions are deselected by resetting bit 2 of MD35035 $MA_SPIND_FUNCTION_MASK.

Reaction: Alarm display.

Remedy: Change deselection block or block search target block to speed control mode (M3, M4, M5, SBCOF). Set bit 2 of MD35035 $MA_SPIND_FUNCTION_MASK to 0.

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 MD35010 $MA_GEAR_STEP_CHANGE_ENABLE[AXn] = 2, the spindle is traversed to the position stored in MD35012 $MA_GEAR_STEP_CHANGE_POSITION[AXn] before the actual gear step change. The required gear step 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
%4 = Gear stage

Definitions: The gear stage required for axis mode has not been installed.
A gear stage has been configured in MR35014 $MA_GEAR_STEP_USED_IN_AXISMODE, in which the spindle is to be in axis mode. This gear stage is checked whenever the spindle is switched into axis mode. The configured gear stage is compared with the gear stage output by the PLC (NC/PLC interface signal V380x 2000.0 - .2 (Actual gear stage A through C)).
This alarm will be output if the gear stages are not the same.

Reaction: Interface signals are set.
Alarm display.

Remedy: Program M70 before the switch to axis mode. The gear stage configured in MD35014 $MA_GEAR_STEP_USED_IN_AXISMODE is then automatically loaded.
No gear stage change is required if the configured gear stage is aleady active. M40 remains active beyond the gear stage change.
Consider MD20094 $MC_SPIND_RIGID_TAPPING_M_NR.

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. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.
If this has been intentionally programmed, the alarm can be suppressed in the cyclic check with position control already enabled by means of MD11410 $MN_SUPPRESS_ALARM_MASK Bit21 = 1.

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

22050 [Channel %1:] Block %3 spindle %2 no transition from speed control mode to position control mode

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

Definitions: - An oriented spindle stop (SPOS/SPOSA) has been programmed or the position control of the spindle was switched on with SPCON but no spindle encoder has been defined.
- When switching on the position control, the spindle speed is greater than the limiting speed of the measuring system.

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

Remedy: Spindle without attached encoder: Any NC language elements requiring the encoder signals must not be used.
Spindle with attached encoder: Enter the number of spindle encoders used in the MD30200 $MA_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: When referencing, the spindle turned through a greater distance than given in the axis-specific MD34060 $MA_REFP_MAX_MARKER_DIST, without receiving a reference mark signal. The check is performed for spindle positioning with SPOS or SPOSA when the spindle has not previously run with speed control (S=...).

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 the MD34060 $MA_REFP_MAX_MARKER_DIST. The value entered states the distance in [mm] or [degrees] between 2 zero markers.

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

22052 [Channel %1:] Block %3 spindle %2 no standstill on block change

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

Definitions: The displayed spindle has been programmed as spindle or as axis even though a positioning operation is still running from the previous block (with SPOSA ... spindle positioning beyond block limits).
Example:
N100 SPOSA [2] = 100
:
N125 S2 = 1000 M2 = 04 ; Error, if spindle S2 from block N100 is still running!

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

Remedy: Before programming the spindle/axis again using the SPOSA instruction, a WAITS command should be activated in order to wait for the programmed spindle position.
Example:
N100 SPOSA [2] = 100
:
N125 WAITS (2)
N126 S2 = 1000 M2 = 04

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

22053 [Channel %1:] Block %3 spindle %2 reference mode not supported

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

Definitions: In the case of SPOS/SPOSA with an absolute encoder, only the referencing mode MD34200 $MA_ENC_REFP_MODE = 2 is supported! SPOS/SPOSA does not support MD34200 $MA_ENC_REFP_MODE = 6 at all!

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

Remedy: Modify setting of MD34200 $MA_ENC_REFP_MODE or change to JOG+REF and then reference.

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

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: Correct NC part program. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.

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

22060 [Channel %1:] Position control expected for axis/spindle %2

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

Definitions: The programmed coupling type (DV, AV) or the programmed function requires position control.

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 marker search velocity is not reached.

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 active spindle speed limitations. Configure a lower zero marker search velocity MD34040 $MA_REFP_VELO_SEARCH_MARKER. Check the tolerance range for the actual velocity MD35150 $MA_SPIND_DES_VELO_TOL. Set a different referencing mode MD34200 $MA_ENC_REFP_MODE != 7.

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

22064 [Channel %1:] Axis %2 reference point approach: zero marker search velocity (MD) is too high

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

Definitions: The configured zero marker search velocity is too high. The encoder limit frequency is exceeded for the active measuring system.

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.
Configure a lower zero marker search velocity MD34040 $MA_REFP_VELO_SEARCH_MARKER. Check the encoder frequency configuration MD36300 $MA_ENC_FREQ_LIMIT and MD36302 $MA_ENC_FREQ_LIMIT_LOW. Set a different referencing mode MD34200 $MA_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 spindle speed is higher than the maximum speed configured in MD35100 $MA_SPIND_VELO_LIMIT plus a tolerance of 10 percent (fixed setting).
The alarm should not occur after correct optimization of the drive actuator and gear configuration.
This alarm can be reconfigured with MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready to operate) to 'BAG not ready'.
Note: Reconfiguring affects all alarms with alarm response 'Chan 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. Check the setup and optimization data of the drive actuator in accordance with the Installation and Start-up Guide and make corrections. Increase the tolerance window in MD35150 $MA_SPIND_DES_VELO_TOL.

Program Continuation: Clear alarm with the RESET key in all channels of this mode group. 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: Please inform the authorized personnel/service department. Provide an interlock in the PLC user program so that no axis stop can be initiated when tapping is active. If the tapping operation is to be terminated under critical machine conditions, the spindle and the axis should be stopped simultaneously if at all possible. Slight differences are then accommodated by the compensating chuck.

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

22250 [Channel %1:] Spindle %2 axis stopped during thread cutting

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

Definitions: The thread cutting axis has been stopped while a thread block was active.
The stop can be caused by VDI signals that cause the feed to be interrupted.

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

Remedy: Please inform the authorized personnel/service department. Check the axis-specific/spindle-specific stop V380x 0004.3 (Spindle stop).

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

22260 [Channel %1:] Spindle %2 thread might be damaged

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

Definitions: When DECODING SINGLE BLOCK has been selected and there is a chain of thread blocks, then machining pauses occur at the block limits until the next block is executed with the new NC Start.
In normal single block mode, the program is stopped by a higher-level logic only at the block boundaries at which no contour distortions or contour errors can occur. With chained thread blocks, this is the last thread block!

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 tapping speed reached

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

Definitions: The following additional parameters are output in order to ascertain the problem:

1. 'Position' (without a preceding axis identifier): The maximum feed of the thread axis is reached at the displayed axis position.
2. 'Master spindle': The position control for the spindle has been changed during thread cutting. This can lead to inaccurate thread cutting.
3. 'Thread axis, velocity': The axis to which the lead refers is displayed. This cannot follow the spindle specifications. The maximum possible axis velocity is shown as a parameter.
The velocity of the thread axis basically depends on:
- The programmed thread pitch
- The programmed thread pitch change and thread length (G34, G35)
- The defined spindle speed (part program, synchronized action)
- The spindle override (path and individual axis overrides are ineffective)

Reaction: Alarm display.

Remedy: Reduce the spindle speed (pitch, pitch change).

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 pitch. The standstill position of the thread axis depends on:
- Programmed thread pitch 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.
The alarm is of the technological type and is output whenever bit 2 in MD11411 $MN_ENABLE_ALARM_MASK is enabled. The HMI softkey 'Technology support' sets and clears this bit in the MD.

Reaction: Alarm display.

Remedy: Modify part program or reset MD11411 $MN_ENABLE_ALARM_MASK bit 2.

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 HMI 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

25000 Axis %1 hardware fault of active encoder

Parameters: %1 = Axis name, spindle number

Definitions: The signals of the currently active position actual value encoder (NC/PLC interface signal V380x 0001.5 = 1 (Position measuring system 1) or V380x 0001.6 = 1 (Position measuring system 2)) are missing, do not have the same phase, or exhibit grounding/short-circuit.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing PowerOn alarm by the Reset alarm 25010.

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. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

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 position actual value encoder are missing, or they are not of the same phase, or they exhibit grounding/short-circuit.
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing PowerOn alarm by the Reset alarm 25011.
MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing Reset alarm by the Cancel alarm 25011.

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. Switch off monitoring with the corresponding interface signal V380x 0001.5 / 1.6 = 0 (Position measuring system 1/2).
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

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

25010 Axis %1 pollution of measuring system

Parameters: %1 = Axis name, spindle number

Definitions: The encoder used for position control sends a contamination signal (only in measuring systems with contamination signal).

The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Reset alarm instead of the Power-on alarm 25000.

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. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

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

25011 Axis %1 pollution of passive encoder

Parameters: %1 = Axis name, spindle number

Definitions: The encoder not used for position control sends a contamination signal (only in measuring systems with contamination signal).
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Reset alarm instead of the Power-on alarm 25001.
MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Cancel alarm instead of the Reset alarm 25001.

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.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

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

25020 Axis %1 zero mark monitoring of active encoder

Parameters: %1 = Axis name, spindle number

Definitions: For PROFIdrive:
The position encoder pulses between 2 zero mark pulses are counted, and the plausibility is assessed (The functionality and possibly the parameteritation of the plausibility check is done on the drive side. Please refer to the relevant drive documentation for details.), and reported in a PROFIdrive message frame (encoder interface) to the control, which then issues the present alarm.
The alarm can be reprogrammed in MD11412 $MN_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.
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.
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 connectors for correct contacting, encoder cable for continuity, and also check for short-circuits and grounding (loose contact?).

2. Encoder pulses: Is the encoder power supply within the tolerance limits?
3. Evaluation electronics: Replace or reconfigure the drive or encoder module used.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING [n] to 0 or 100 (n = encoder number: 1, 2).

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

25021 Axis %1 zero mark monitoring of passive encoder

Parameters: %1 = Axis name, spindle number

Definitions: Monitoring relates to the encoder that is not used by the position control. (NC-PLC interface signal V380x 0001.5 = 0 (Position measuring system 1) or V380x 0001.6 = 0 (Position measuring system 2))
More detailed explanations are similar to those for alarm 25020.

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 not used for position control. The actual value branch must therefore be checked:
1. Transmission path: Check the actual-value connectors for correct contacting, encoder cable for continuity, and also check for short-circuits and grounding (loose contact?).
2. Encoder pulses: Is the encoder power supply within the tolerance limits?
3. Evaluation electronics: Replace or reconfigure the drive or encoder module used.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 0 or 100 (n = encoder number: 1, 2).

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

25030 Axis %1 actual velocity alarm limit

Parameters: %1 = Axis name, spindle number

Definitions: If the axis has at least one active encoder, then the actual speed of the axis is cyclically checked in the IPO cycle. If there are no errors, the actual velocity can never become greater than specified in the axis-specific MD36200 $MA_AX_VELO_LIMIT (threshold for velocity monitoring). This threshold value in [mm/min, rev/min] is input by an amount that is about 5 to 10% greater than that which can occur at maximum traversing velocity. Drive errors can result in the velocity being exceeded and the alarm is then triggered.
The alarm can be reprogrammed in the MD11412 $MN_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 the speed setpoint cable (bus cable).
- Check the actual values and direction of position control.
- Change the position control direction if the axis rotates uncontrollably -> axis-specific MD32110 $MA_ENC_FEEDBACK_POL [n] = < -1, 0, 1 >.
- Increase the monitoring limit value in MD 36200 $MA_AX_VELO_LIMIT.

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

25040 Axis %1 standstill monitoring

Parameters: %1 = Axis name, spindle number

Definitions: The NC monitors to ensure that the position is held at zero speed. Monitoring is started after a time that can be set for a specific axis in the MD36040 $MA_STANDSTILL_DELAY_TIME after interpolation has ended. A constant check is made to determine whether the axis remains within the tolerance range given in MD36030 $MA_STANDSTILL_POS_TOL.
The following cases are possible:
1. The NC/PLC interface signal V380x 0002.1 (Servo enable) is zero because the axis has jammed mechanically. Due to mechanical influences (e.g. high machining pressure), the axis is pushed outside the permissible position tolerance.

2. With closed position control loop (without jamming) - NC/PLC interface signal V380x 0002.1 (Servo enable) is "1" - the axis is pushed away from its position by mechanical forces with a small gain in the position control loop.
The alarm can be reprogrammed in the MD11412 $MN_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 MD36040 $MA_STANDSTILL_DELAY_TIME and MD36030 $MA_STANDSTILL_POS_TOL; increase if necessary. The value must be greater than the machine data "Exact stop - coarse" (MD36000 $MA_STOP_LIMIT_COARSE).
- Estimate machining forces and reduce if necessary by setting a lower feed or a higher rotational speed.
- Increase clamping pressure.
- Increase the gain in the position control loop by improved optimization (Kv factor MD32200 $MA_POSCTRL_GAIN, SIMODRIVE611D drive).

Program Continuation: 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 NCK calculates for each interpolation point (setpoint) of an axis the actual value that should result based on an internal model. If this calculated actual value and the true machine actual value differ by a larger amount than given in the MD36400 $MA_CONTOUR_TOL, then the program is aborted and the alarm message is issued.
The alarm can be reprogrammed in the MD11412 $MN_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 whether the tolerance value set in MD36400 $MA_CONTOUR_TOL is too small.
- Check optimization of the position controller (Kv factor in the MD32200 $MA_POSCTRL_GAIN) to establish whether the axis follows the given setpoint without overshooting. Otherwise, the speed controller optimization must be improved or the Kv servo gain factor must be reduced.
- Improvement of speed controller optimization
- Check the mechanics (smooth running, inertial masses).

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

25060 Axis %1 speed setpoint limitation

Parameters: %1 = Axis name, spindle number

Definitions: The speed setpoint has exceeded its upper limit for a longer period than allowed.
The maximum speed setpoint is limited to a certain percentage by the axis-specific MD36210 $MA_CTRLOUT_LIMIT. The input value of 100% corresponds to the rated speed of the motor and hence the rapid traverse velocity.
For SINAMICS: Drive parameter p1082 also has a limiting effect.
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 MD36220 $MA_CTRLOUT_LIMIT_TIME. The setpoint is limited during this time to the maximum value that has been set in (MD36210 $MA_CTRLOUT_LIMIT).

The alarm can be reprogrammed in the MD11412 $MN_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. This alarm should not occur if the drive controller has been set correctly and the machining conditions are those that normally prevail.
- Check actual values: Local sluggishness of the carriage, speed dip by torque surge due to contact with workpiece/tool, travel against fixed obstacle, etc.
- Check direction of position control: Does the axis continue to rotate without control?

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

25070 Axis %1 drift value too large

Parameters: %1 = Axis name, spindle number

Definitions: Only with analog drives!
The permissible maximum value of drift (internal, integrated drift value of automatic drift compensation) was exceeded during the last compensation operation. The permissible maximum value is defined in the axis-specific MD36710 $MA_DRIFT_LIMIT. The drift value itself is not limited.
Automatic drift compensation: MD36700 $MA_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 automatically compensated to zero by slowly integrating an internal drift value.
Drift compensation by hand: MD36700 $MA_DRIFT_ENABLE=0
A static offset can be added to the speed setpoint in the MD36720 $MA_DRIFT_VALUE. This is not included in the drift monitoring because it acts like a voltage zero offset.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Adjust the drift compensation with the automatic drift compensation switched off at the drive until the position lag is approximately zero. Then reactivate the automatic drift compensation in order to balance out the dynamic drift changes (effects of heating up).

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

25080 Axis %1 positioning monitoring

Parameters: %1 = Axis name, spindle number

Definitions: For blocks in which "exact stop" is effective, the axis must have reached the exact stop window after the positioning time given in the axis-specific MD36020 $MA_POSITIONING_TIME.
Exact stop coarse: MD36000 $MA_STOP_LIMIT_COARSE
Exact stop fine: MD36010 $MA_STOP_LIMIT_FINE
The alarm can be reprogrammed in the MD11412 $MN_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 whether the exact stop limits (coarse and fine) correspond to the dynamic possibilities of the axis, otherwise increase them, if necessary in connection with the positioning time set in MD36020 $MA_POSITIONING_TIME.
Check speed controller/position controller optimization; select highest possible gain.
Check setting of Kv factor (MD32200 $MA_POSCTRL_GAIN) and increase, if required.

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

25105 Axis %1 measuring systems differ considerably

Parameters: %1 = Axis name, spindle number

Definitions: The two measuring systems differ considerably, i.e. the cyclically monitored actual value difference between the two measuring systems is greater than the associated tolerance value set in the machine data MD36510 $MA_ENC_DIFF_TOL. This can only occur when both measuring systems are active (MD30200 $MA_NUM_ENCS = 2) and referenced. The alarm can be reprogrammed in the MD11412 $MN_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 for the active, selected encoders. Check the machine data relating to encoder (MD36510 $MA_ENC_DIFF_TOL) tolerance.

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

25110 Axis %1 selected encoder not available

Parameters: %1 = Axis name, spindle number

Definitions: The selected encoder does not correspond to the maximum number of encoders in the axis-specific MD30200 $MA_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 MD30200 $MA_NUM_ENCS ("Number of encoders").
Input value 0: Axis without encoder -> e.g. spindle
Input value 1: Axis with encoder -> default setting
Input value 2: Axis with 2 encoders -> e.g. direct and indirect measuring system

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

25200 Axis %1 requested set of parameters invalid

Parameters: %1 = Axis name, spindle number

Definitions: A new parameter set has been requested for the positioning control. The number of this parameter set is beyond the permissible limit (8 parameter sets: 0 ... 7 available).

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-specific/spindle-specific interface signals <Regler-Parametersatz1A-C/> (Select parameter set servo A, B, C).
One parameter set includes the following machine data:
- MD31050 $MA_DRIVE_AX_RATIO_DENOM [n]
- MD31060 $MA_DRIVE_AX_RATIO_NUMERA [n]
- MD32200 $MA_POSCTRL_GAIN [n]
- MD32800 $MA_EQUIV_CURRCTRL_TIME [n]
- MD32810 $MA_EQUIV_SPEEDCTRL_TIME [n]
- MD32910 $MA_DYN_MATCH_TIME [n]
- MD36200 $MA_AX_VELO_LIMIT [n]

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

25201 Axis %1 drive fault

Parameters: %1 = Axis name, spindle number

Definitions: For PROFIdrive:
The drive signals a serious fault which prevents the drive from being ready. The exact cause of the fault can be found by evaluating the additionally output drive alarms (It may be necessary to activate these diagnostic alarms by parameterizing the MDs DRIVE_FUNCTION_MASK, PROFIBUS_ALARM_ACCESS etc):
Alarms 380500 and 380501 (or the corresponding alarm numbers implemented on the HMI side).
The alarm can be reprogrammed in the MD11412 $MN_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: Evaluation of the drive alarms listed above.

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

25202 Axis %1 waiting for drive

Parameters: %1 = Axis name, spindle number

Definitions: Drive group error (self-clearing).

Reaction: Interface signals are set.
Alarm display.

Remedy: For PROFIdrive only:
Wait for the drive. This alarm reveals similar problems to alarm 25201 (see that alarm). It is continuously active during power-up if the drive does not communicate (e.g. if the PROFIBUS connector has fallen out). 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 MD36050 $MA_CLAMP_POS_TOL.
Clamping an axis is activated with the axis-specific interface signal V380x 0002.3 (Clamping process active).
The alarm can be reprogrammed in the MD11412 $MN_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: Determine the position deviation to the setpoint position and, depending on the results, either increase the permissible tolerance in the MD or mechanically improve the clamping (e.g. increase clamping pressure).

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

26001 Axis %1 parameterization error: friction compensation

Parameters: %1 = Axis name, spindle number

Definitions: The parameterization of the adaptation characteristic in the quadrant error compensation is not allowed because acceleration value 2 (MD32560 $MA_FRICT_COMP_ACCEL2 is not between acceleration value 1 (MD32550 $MA_FRICT_COMP_ACCEL1) and acceleration value 3 (MD32570 $MA_FRICT_COMP_ACCEL3).
The alarm can be reprogrammed in the MD11412 $MN_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 the setting parameters of the quadrant error compensation (friction compensation), if necessary switch off the compensation with MD32500 $MA_FRICT_COMP_ENABLE.

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

26002 Axis %1 encoder %2 parameterization error: number of encoder marks

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

Definitions: 1. Rotary measuring system (MD31000 $MA_ENC_IS_LINEAR[]==FALSE)
The number of encoder marks set in MD31020 $MA_ENC_RESOL[] does not correspond to the value in the drive machine data (SIMODRIVE 611D: MD1005 $MD_ENC_RESOL_MOTOR; PROFIdrive: p979) or zero has been entered in one of the two machine data.
2. Absolute measuring system with EnDat interface (MD5790 $MD_ENC_TYPE[]==4)
On absolute encoders, the resolution of the incremental and absolute tracks supplied by the drive is also checked for consistency.
For PROFIdrive drives:
Compare drive parameter p979 (and possibly other internal drive, manufacture-specific parameters stated in the relevant drive documentation)

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.
Adjust machine data.
For absolute encoders, possibly pending drive alarms indicating encoder problems should be evaluated. They could be the cause of incorrect entries of MD1022 $MD_ENC_ABS_RESOL_MOTOR/MD1032 $MD_ENC_ABS_RESOL_DIRECT, which the drive reads out of the encoder itself.

Program Continuation: Switch control OFF - ON.

26003 Axis %1 parameterization error: lead screw pitch

Parameters: %1 = Axis name, spindle number

Definitions: The pitch of the ballscrew/trapezoidal leadscrew set in the axis-specific MD31030 $MA_LEADSCREW_PITCH is zero.
The alarm can be reprogrammed in the MD11412 $MN_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: Determine the leadscrew pitch (specify the machine manufacturer or pitch measurement with spindle cover removed) and enter it in the MD31030: $MA_LEADSCREW_PITCH (mostly 10 or 5 mm/rev.).

Program Continuation: Switch control OFF - ON.

26004 Axis %1 encoder %2 parameterization error: grid point distance with linear encoders

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

Definitions: The scale division of the linear scale set in the axis-specific MD31010 $MA_ENC_GRID_POINT_DIST is zero or differs from the corresponding drive parameters. For a better understanding of the interrelations see the explanations for alarm 26002, which refer to rotatory encoders.
The alarm can be reprogrammed in the MD11412 $MN_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.
Enter the encoder grid point distance according to the data given by the machine (or measuring device) manufacturer in the MD31010 $MA_ENC_GRID_POINT_DIST.

Program Continuation: Switch control OFF - ON.

26005 Axis %1 parameterization error: output rating

Parameters: %1 = Axis name, spindle number

Definitions: For analog drives:
The output evaluation of the analog speed setpoint set in the MD32250 $MA_RATED_OUTVAL or in MD 32260 $MA_RATED_VELO is zero.
For PROFIdrive drives: (ADI4, SIMODRIVE 611U, SINAMICS):
The effective output evaluation of the speed setpoint interface is zero:
-
b. The corresponding drive-side standardizing parameter is zero, invalid or unreadable/unavailable although an automatic interface scaling adjustment has been selected on account of MD32250 $MA_RATED_OUTVAL=0.
The drive parameter defining the standard is not determined by PROFIdrive, but is specific to the manufacturer (see the relevant drive documentation: For SIMODRIVE 611U: p880; for SINAMICS: p2000).
The alarm can be reprogrammed in the MD11412 $MN_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. The nominal output voltage in [%] of the maximum setpoint value (10 V) is entered in the MD32250 $MA_RATED_OUTVAL, at which the rated motor speed in [degrees/s] is to be reached (MD32260 $MA_RATED_VELO).

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

26006 Axis %1 encoder %2 encoder type/output type %3 not possible

Parameters: %1 = Axis name, spindle number
%2 = Encoder number
%3 = Encoder type/output type

Definitions: Not every encoder type or output type can be used with every control or drive variant.
Permissible settings:
MD30240 $MA_ENC_TYPE
= 0 Simulation (always permissible)
= 1 Raw signal incremental encoder (SIMODRIVE 611D and PROFIdrive)
= 4 Absolute encoder (EnDat with SIMODRIVE 611D; all drive-side absolute encoders supported by PROFIdrive)
MD30130 $MA_CTRLOUT_TYPE
= 0 Simulation
= 1 Standard (SIMODRIVE 611D and PROFIdrive drives)
The alarm can be reprogrammed in the MD11412 $MN_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 MD30240 $MA_ENC_TYPE and/or MD30130 $MA_CTRLOUT_TYPE and make the necessary corrections.

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: Machine data includes a value that is not valid.

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: Machine data includes a value that is not valid.

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 includes a value that is not valid.

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: 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

Definitions: Machine data includes a value that is not valid.

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: 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: The same setpoint assignment has been allocated more than once.
MD30110 $MA_CTRLOUT_MODULE_NR contains the same value for different axes.
PROFIdrive: The stated MDs contain the same values for different axes, or different entries in $MN_DRIVE_LOGIC_ADDRESS contain the same values.

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 dual assignment of the setpoint by correcting MD30110 $MA_CTRLOUT_MODULE_NR. Also check the selected bus type MD30100 $MA_CTRLOUT_SEGMENT_NR.

Program Continuation: Switch control OFF - ON.

26019 Axis %1 encoder %2 measurement not possible with this controller module

Parameters: %1 = NC axis number
%2 = Encoder number

Definitions: If the MD MD13100 $MN_DRIVE_DIAGNOSIS[8] contains a value not equal to zero, then the control has found at least one control module which does not support measuring. Measuring was programmed from the part program for the associated axis.

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 such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis. Otherwise, exchange the controller module for one that supports measuring. See MD13100 $MN_DRIVE_DIAGNOSIS[8].

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

26022 Axis %1 encoder %2 measurement with simulated encoder not possible

Parameters: %1 = NC axis number
%2 = Encoder number

Definitions: Alarm occurs on the control when a measurement was made without the encoder hardware (simulated encoder).

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 such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis.
- Ensure that measurement is not taking place with simulated encoders (MD30240 $MA_ENC_TYPE).

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

26024 Axis %1 machine data %2 value changed

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

Definitions: The machine data contains an invalid value and therefore has 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 was therefore changed by the software internally to a valid value.

Reaction: Alarm display.

Remedy: Check MD.

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

26030 Axis %1 encoder %2 absolute position lost

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

Definitions: The absolute position of the absolute encoder has become invalid because
- on changing parameter block a changed gear stage ratio was identified between encoder and processing or
- the encoder has been replaced (the absolute encoder's serial number has changed, see MD34230 $MA_ENC_SERIAL_NUMBER, and drive-specific parameters).

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. Rereferencing/resynchronization of the absolute encoder; attach absolute encoder on the load side and configure correctly (e.g. MD 31040 $MA_ENC_IS_DIRECT).
If an absolute encoder with a serial number is replaced, the drive BOT file for this drive must be saved (because of the new serial number).

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

26031 Axis %1 configuration error master-slave

Parameters: %1 = Axis name, spindle number

Definitions: The alarm is output when the same machine axis has been configured as a master and a slave axis. Each of the axes in the master/slave link can be operated either as master or slave.

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: - Check machine data for all linked axes and correct if necessary:
- MD 37250 $MA_MS_ASSIGN_MASTER_SPEED_CMD
- MD 37252 $MA_MS_ASSIGN_MASTER_TORQUE_CTR.

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

26032 [Channel %1:] Axis %2 master-slave not configured

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

Definitions: The master-slave coupling could not be activated because of incomplete configuration.

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

Remedy: Check the current configuration of the master-slave coupling.
The configuration can be modified via the MASLDEF instruction or the machine data MD37250 $MA_MS_ASSIGN_MASTER_SPEED_CMD and MD37252 $MA_MS_ASSIGN_MASTER_TORQUE_CTR.

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

26050 Axis %1 parameter set change from %2 to %3 not possible

Parameters: %1 = Axis name, spindle number
%2 = Index: current parameter block
%3 = Index: new parameter block

Definitions: The parameter block change cannot be performed without jumps. This is due to the content of the parameter block to be switched on, e.g. different load gear factors.

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: In the following cases, the parameter block change is carried out via MD31060 $MA_DRIVE_AX_RATIO_NUMERA and MD31050 $MA_DRIVE_AX_RATIO_DENOM without an alarm, even with different load gear ratio settings:

1. If no position control is active (e.g. in follow-up mode or if spindle is in speed-controlled mode).
2. For position control with the direct encoder.
3. For position control with the indirect encoder (the calculated load position difference must not exceed the value indicated in MD36500 $MA_ENC_CHANGE_TOL).

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

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 also 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 MD36400 $MA_CONTOUR_TOL setting was over-sensitive).

Reaction: Alarm display.

Remedy: - If the alarm occurred in a block with auxiliary function output during the movement: from SW 5.1 or higher, increase machine MD10110 $MN_PLC_CYCLE_TIME_AVERAGE or
- Program G09 in the block indicated in the message to allow the path interpolation to stop as planned.

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

26070 [Channel %1:] Axis %2 cannot be controlled by the PLC, max. number exceeded

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

Definitions: An attempt was made to define more axes than allowed as PLC-controlled axes.

Reaction: Interface signals are set.
Alarm display.

Remedy: Check the option 'Number of PLC-controlled axes' and correct if necessary or reduce the number of requests for 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: 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.

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 processing only, if the axis is in READY state.

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

Remedy: Reset VDI interface signal "PLC controls axis", then activate "Axial reset" and repeat process.

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

26100 Axis %1 drive %2 sign of life missing

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

Definitions: Special case: The output of drive number=0 indicates that a computing timeout occurred on the IPO level (see also alarm 4240)

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: Restart drive, check drive software.

Program Continuation: Switch control OFF - ON.

26101 Axis %1 drive %2 communication failure

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

Definitions: For PROFIdrive only:
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 the bus configuration.
- Check the interface (connector removed, option module inactive, etc.).

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

26102 Axis %1 drive %2 sign of life missing

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

Definitions: For PROFIdrive only:
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 the cycle settings.
- Increase the cycle time if necessary.
- Power-up the drive again.
- Check drive software.

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

26105 Drive of axis %1 not found

Parameters: %1 = Axis name, spindle number

Definitions: For PROFIdrive only:
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-Type-2000.

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:
- MD30130 $MA_CTRLOUT_TYPE not equal to 0 as a result of an oversight; the drive should actually be simulated (= 0).
- MD30110 $MA_CTRLOUT_MODULE_NR entered incorrectly, i.e. the logical drive numbers were exchanged and an invalid value is stored for this drive in MD13050 $MN_DRIVE_LOGIC_ADDRESS (see 3.) or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- MD13050 $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the Profibus (i.e. the values are not in SDB-Type-2000) or different addresses were selected for the input and output slots of the drive in the Profibus configuration.

Program Continuation: Switch control OFF - ON.

26106 Encoder %2 of axis %1 not found

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

Definitions: For PROFIdrive only:
The encoder configured for the specified axis could not be found. For example, a PROFIBUS slave was configured on the NC but it is not contained in the SDB or defective hardware was reported for it.

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 $MA_ENC_TYPE not equal to 0 as a result of an oversight; the encoder should actually be simulated (= 0).
- MD 30220 $MA_ENC_MODULE_NR entered incorrectly, i.e. the logical drive numbers were transposed and an invalid value is stored for this drive in MD 13050 $MN_DRIVE_LOGIC_ADDRESS (see next paragraph), or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- MD 13050 $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the Profibus (i.e. the values are not in SDB Type 2000) or different addresses were selected for the input and output slots of the drive in the Profibus configuration.
- A fatal encoder error was detected during the selection of the encoder (encoder defective, removed), so that park status cannot be left (in such a case, this alarm is triggered instead of alarm 25000/25001 - please refer to them for other possible causes of the error).

Program Continuation: Switch control OFF - ON.

29033 **[Channel %1:] Axis exchange of axis %2 not possible, PLC axis movement not yet completed**

Parameters: %1 = Channel number
%2 = Axis

Definitions: A PLC axis has not yet reached its end position and cannot be returned to a channel or neutralized.

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

2.2 Cycle alarms

61000 [Channel %1:] Block %2: No tool offset active

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

Definitions: Alarm triggered by following cycles: LONGHOLE, SLOT1, SLOT2, POCKET1 through POCKET4, CYCLE71, CYCLE72, CYCLE90, CYCLE93 through CYCLE96.

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

Remedy: D-correction must be programmed prior to cycle call.

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

61001 [Channel %1:] Block %2: Thread pitch incorrectly defined

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

Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840, CYCLE96, CYCLE97.

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

Remedy: Check parameter for the thread size or setting for the pitch (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: %1 = Channel number
%2 = Block number, label

Definitions: The value of the VARI parameter for the machining has been incorrectly specified. Alarm triggered by following cycles: SLOT1, SLOT2, POCKET1 to POCKET4, CYCLE71, CYCLE72, CYCLE76, CYCLE77, CYCLE93, CYCLE95, CYCLE97, CYCLE98.

Remedy: Modify VARI parameter.

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

61003 [Channel %1:] Block %2: No feed programmed in cycle

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

Definitions: The parameter for the feed has been incorrectly specified. Alarm triggered by following cycles: CYCLE71, CYCLE72.

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

Remedy: Modify feed parameter.

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

61005 [Channel %1:] Block %2: 3rd geometry axis not available

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

Definitions: With an application on the lathe with no Y-axis in the G18 plane. Alarm triggered by following cycle: CYCLE86.

Remedy: Check parameter on cycle call.

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

61008 [Channel %1:] Block %2: No tool active

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

Definitions: Alarm triggered by following cycles:

Remedy: Select a tool.

61009 [Channel %1:] Block %2: Active tool number = 0

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

Definitions: No tool (T) has been programmed before the cycle call. Alarm triggered by following cycles: CYCLE71, CYCLE72.

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

Alarm display.

Remedy: Program tool (T).

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

61010 [Channel %1:] Block %2: Finishing allowance too large

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

Definitions: The finishing allowance for the base is greater than the total depth. Alarm triggered by following cycle: CYCLE72.

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

Remedy: Reduce finishing allowance.

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

61011 [Channel %1:] Block %2: Scaling not permissible

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

Definitions: A scale factor is active which is illegal for this cycle. Alarm triggered by following cycles: CYCLE71, CYCLE72.

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

Remedy: Modify scale factor.

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

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

61014 [Channel %1:] Block %2: Return plane exceeded

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

Definitions: Alarm triggered by following cycles: CYCLE72.

Remedy: Check parameter RTP.

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

61016 [Channel %1:] Block %2: System frame for cycles missing

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

Definitions: All measuring cycles can trigger this alarm.

Remedy: Set MD 28082: MM_SYSTEM_FRAME_MASK, Bit 5=1.

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.

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

61024 [Channel %1:] Block %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

61101 [Channel %1:] Block %2: Reference plane incorrectly defined

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

Definitions: Alarm triggered by following cycles: CYCLE71, CYCLE72, CYCLE81 to CYCLE90, CYCLE840, SLOT1, SLOT2, POCKET1 to POCKET4, LONGHOLE.

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

Remedy: With the relative specification of the depth, either different values for the reference plane and the retraction plane must be selected or an absolute value must be specified for the depth.

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

61102 [Channel %1:] Block %2: No spindle direction programmed

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

Definitions: Alarm triggered by following cycles: CYCLE86, CYCLE87, CYCLE88, CYCLE840, POCKET3, POCKET4.

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

Definitions: No value for the number of holes has been programmed. Alarm triggered by following cycles: HOLES1, HOLES2.

Remedy: Check parameter NUM.

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

61104 [Channel %1:] Block %2: Contour violation of slots

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

Definitions: Incorrect parameterization of the milling pattern in the parameters which define the position of the slots/elongated holes on the circle and their form. Alarm triggered by following cycles: SLOT1, SLOT2, LONGHOLE.

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

61105 [Channel %1:] Block %2: Milling cutter radius too large

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

Definitions: The diameter of the cutter used is too large for the form to be machined. Alarm triggered by following cycles: SLOT1, SLOT2, POCKET1 to POCKET4, LONGHOLE, CYCLE90.

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

Remedy: Either a tool with a smaller radius has to be used or the contour must be modified.

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

61106 [Channel %1:] Block %2: Number of or distance between circular elements

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

Definitions: Incorrect parameterization of NUM or INDA. The layout of the circle elements within a full circle is not possible. Alarm triggered by following cycles: HOLES2, LONGHOLE, SLOT1, SLOT2.

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

Remedy: Correct parameterization.

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

61107 [Channel %1:] Block %2: First drilling depth incorrectly defined

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

Definitions: First drilling depth is in the opposite direction to the total drilling depth. Alarm triggered by following cycle: CYCLE83.

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

Remedy: Modify 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: %1 = Channel number
%2 = Block number, label

Definitions: The parameters _RAD1 and _DP for defining the path for the depth infeed have been incorrectly specified. Alarm triggered by following cycles: POCKET3, POCKET4.

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

Remedy: Modify parameter.

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

61109 [Channel %1:] Block %2: Parameter _CDIR incorrectly defined

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

Definitions: The value of the parameter for the cutting direction _CDIR has been incorrectly defined. Alarm triggered by following cycles: POCKET3, POCKET4.

Remedy: Change parameter _CDIR.

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

61110 [Channel %1:] Block %2: Finishing allowance at bottom > depth infeed

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

Definitions: The finishing allowance at the base has been specified greater than the maximum depth infeed. Alarm triggered by following cycles: POCKET3, POCKET4.

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

Remedy: Either reduce finishing allowance or increase depth infeed.

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

61111 [Channel %1:] Block %2: Infeed width > Tool diameter

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

Definitions: The programmed infeed width is greater than the diameter of the active tool. Alarm triggered by following cycles: CYCLE71, POCKET3, POCKET4.

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

Remedy: Infeed width must be reduced.

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

61112 [Channel %1:] Block %2: Tool radius negative

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

Definitions: The radius of the active tool is negative. This is illegal. Alarm triggered by following cycles: CYCLE72, CYCLE76, CYCLE77, CYCLE90.

Remedy: Change the tool radius.

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

61113 [Channel %1:] Block %2: Parameter _CRAD too large for corner radius

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

Definitions: The parameter for the corner radius _CRAD has been specified too large. Alarm triggered by following cycle: POCKET3.

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

Remedy: Parameter must be reduced.

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

61114 [Channel %1:] Block %2: Machining direction G41/G42 incorrectly defined

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

Definitions: The machining direction of the cutter radius compensation G41/G42 has been incorrectly selected. Alarm triggered by following cycle: CYCLE72.

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

Remedy: Change machining direction.

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

61115 [Channel %1:] Block %2: Approach or retract mode(straight / circle / plane / space) incorrectly defined

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

Definitions: The approach or retract mode to/from the contour has been incorrectly defined. Alarm triggered by following cycle: CYCLE72.

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

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

Definitions: The approach or retract path has been specified with zero. Alarm triggered by following cycle: CYCLE72.

Reaction: Interpreter stop
NC Start disable in this channel.

Interface signals are set.
Alarm display.

Remedy: Check parameter _LP1 or _LP2.

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

61117 [Channel %1:] Block %2: Active tool radius <= 0

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

Definitions: The radius of the acive tool is negative or zero. Alarm triggered by following cycles: CYCLE71, POCKET3, POCKET4.

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

Remedy: Modify radius.

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

61118 [Channel %1:] Block %2: Length or width = 0

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

Definitions: The length or width of the milling area is illegal. Alarm triggered by following cycle: CYCLE71.

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

Remedy: Check parameters _LENG and _WID.

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

61119 [Channel %1:] Block %2: Nominal or core diameter 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

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

Definitions: Alarm triggered by following cycle: CYCLE71.

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

Remedy: With active simulation without tool, a value for the infeed width _MIDA must always be programmed.

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

61125 [Channel %1:] Block %2: Technology selection in parameter _TECHNO incorrectly defined

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

Definitions: Alarm triggered by following cycles: CYCLE84, CYCLE840.

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.

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

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

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

61150 [Channel %1:] Block %2: Tool cannot be aligned --> error code: %4

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

Definitions: Alarm triggered by following cycle: CYCLE800.

Remedy: Error code:
A -> only new swivel plane permitted, see parameter _ST

61151 [Channel %1:] Block %2: Orientation of tool not possible --> error code: %4

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

Definitions: Alarm triggered by following cycle: CYCLE800.

Remedy: Error code:
A -> only additive swivel plane permitted, see parameter _ST

61152 [Channel %1:] Block %2: B axis kinematics (turning technology) either not or incorrectly set up in Start-up of swivel cycle --> error code: %4

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

Definitions: Alarm triggered by following cycle: CYCLE800.

Remedy: Error code:
A123 -> B axis is not an automatic rotary axis under ShopTurn (123 corresponds to parameter _TCBA)
B123 -> B axis is not activated in start-up swiveling (kinematics)
(123 corresponds to $TC_CARR37[n], n ... number of the swivel data record)

61153 [Channel %1:] Block %2: No 'Rotary axes direct' swivel mode possible --> error code: %4

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

Definitions: Alarm triggered by following cycle: CYCLE800.

Remedy: Error code:
A -> No tool or cutting edge (D1..) active

61154 [Channel %1:] Block %2: Final depth wrongly programmed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Input of end depth possible only absolutely or incrementally

61155 [Channel %1:] Block %2: Unit for plane infeed wrongly programmed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Unit for plane infeed possible only in mm or % of tool diameter

61156 [Channel %1:] Block %2: Depth calculation wrongly programmed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Depth calculation possible only with or without SDIS

61157 [Channel %1:] Block %2: Reference point wrongly programmed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Check reference point in screen form, input only -X, centred or +X

61158 [Channel %1:] Block %2: Machining plane wrongly programmed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Check machining plane (G17, G18 or G19)

61159 [Channel %1:] Block %2: Machining plane on cycle call differs from the one in the position pattern

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Adjust the machining plane on cycle call to the machining plane in the position pattern.

61160 [Channel %1:] Block %2: Residual material remains stationary, reduce plane infeed

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

Definitions: The alarm is triggered by the following cycle: CYCLE899

Remedy: Reduce plane infeed or slot width, or use milling cutter with larger diameter

61161 [Channel %1:] Block %2: Centering diameter or tool parameter (diameter, tip angle) are incorrect

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

Definitions: The alarm is triggered by the following cycle: CYCLE81

Remedy: - Diameter of centering with tip angle of active tool not possible
- Entered workpiece diameter, tool diameter or tip angle of tool incorrect
- Diameter of tool only has to be entered if centering is to be on workpiece diameter.

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

61162 [Channel %1:] Block %2: Tool parameter diameter or tip angle incorrect

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

Definitions: The alarm is triggered by the following cycle: CYCLE81

Remedy: - The tool parameter diameter or tip angle must be greater than zero
- Tip angle must be less than 180°

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

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.

Remedy: Enter another character.

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

61180 [Channel %1:] Block %2: no name assigned to swivel data block, although MD $MN_MM_NUM_TOOL_CARRIER > 1

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

Definitions: Although there are several swivel data blocks, no unique names have been assigned. Alarm triggered by following cycles: CYCLE800.

Remedy: Assign unique names for swivel data blocks.

61181 [Channel %1:] Block %2: Insufficient NCK software version (TOOLCARRIER functionality missing)

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

Definitions: Swivelling is not possible with the current NCK software version. Alarm triggered by following cycles: CYCLE800.

Remedy: Upgrade NCK software; functionality TOOLCARRIER available in NCU 6.3xx and higher.

61182 [Channel %1:] Block %2: Name of swivel data record unknown

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

Definitions: the specified name of the swivel data block is unknown. Alarm triggered by following cycles: CYCLE800.

Remedy: Check the name of the swivel data block.

61183 [Channel %1:] Block %2: Clearance mode _FR outside of value range 0..2.

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

Definitions: The retraction mode value lies outside of the valid range. Alarm triggered by following cycles: CYCLE800.

Remedy: Check installation and start-up of swivel cycle CYCLE800 -> retraction parameter _FR.

61184 [Channel %1:] Block %2: No solution possible with current input angle values

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

Definitions: The surface defined via the input angle cannot be processed with the machine. Alarm triggered by following cycles: CYCLE800.

Remedy: -Check the angle entered for the swiveling of the machining plane.
-Parameter _MODE coding incorrect, e.g. rotation axis-wise YXY

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

61186 [Channel %1:] Block %2: Invalid rotary axis vectors --> Start-up check swivel cycle CYCLE800

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

Definitions: No or incorrect entry for rotary axis vector V1 or V2. Alarm triggered by following cycles: CYCLE800.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.
Check rotary axis vectors V1 and V2.

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

61188 [Channel %1:] Block %2: No axis name agreed for 1st rotary axis -> Start-up check CYCLE800

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

Definitions: No axis name was specified for the 1st rotary axis. Alarm triggered by following cycles: CYCLE800.

Remedy: Check installation and start-up of the swivel cycle CYCLE800.
No entry under rotary axis 1 identifier.

61189 [Channel %1:] Block %2: invalid rotary axis positions

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

Definitions: Alarm triggered by following cycles: CYCLE800.

Remedy: Swivel in JOG, swivel mode direct, check position of rotary axes or start-up of swivel cycle CYCLE800 rotary axes, check area of angles.

61190 [Channel %1:] Block %2: unable to retract in tool direction --> error code: %4

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

Definitions: The alarm is triggered by the following cycles: CYCLE800.

Remedy: Error code:
A=0xxx -> Parameter CYCLE800 _FR incorrect or retraction variant not created in start-up CYCLE800.
A=1xxx -> No applicats available $P_AXN3
A=2xxx -> Maximum retraction path incorrect, see GUD _TC_P[8]
A=3xxx -> Incremental retraction path incorrect, see GUD _TC_P[8
A=4xxx -> Retraction in tool direction, NC function CALCPOSI reports error
No reference for tool axis (e.g. Z with G17) approached

B = Input parameter _FR*100
CD = Start-up parameter $P_TCARR37[] (7th, 8th decimal place)
See table Coding retraction modes

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

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

61196 [Channel %1:] Block %2: no swiveling in JOG --> 5 axis transformation and TCARR simultaneously activated

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

Definitions: 5-axis transformation and TOOLCARRIER activated at the same time.
Alarm triggered by following cycles: CYCLE800.

Remedy: 5-axis transformation and TOOLCARRIER activated at the same time.

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.

Remedy: --

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

61200 [Channel %1:] Block %2: Too many elements in machining block

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

Definitions: The machining block contains too many elements.
Alarm triggered by following cycles: CYCLE76, CYCLE77, E_CALL, E_DR, E_DR_BGF, E_DR_BOR, E_DR_O1, E_DR_PEC, E_DR_REA, E_DR_SIN, E_DR_TAP, E_MI_TR, E_PI_CIR, E_PI_REC, E_PO_CIR, E_PO_REC, E_PS_CIR, E_PS_FRA, E_PS_HIN, E_PS_MRX, E_PS_POL, E_PS_ROW, E_PS_SEQ, E_PS_XYA, E_SL_LON, F_DR, F_DR_PEC, F_DR_REA, F_DR_SIN, F_DR_TAP, F_MI_TR, F_PI_CIR, F_PI_REC, F_PO_CIR, F_PO_REC, F_PS_CIR, F_PS_MRX, F_PS_ROW, F_PS_SEQ, F_SL_LON

Remedy: Check the machining block, delete some elements, if required.

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

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

61213 [Channel %1:] Block %2: Circle radius too small

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

Definitions: The programmed circle radius is too small.
Alarm triggered by following cycles: CYCLE77, E_CR_HEL, E_PI_CIR, E_PO_CIR, E_PO_REC, F_PI_CIR, F_PO_CIR, F_PO_REC

Remedy: Correct the circle radius, center point or end point.

61214 [Channel %1:] Block %2: No pitch programmed

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

Definitions: No thread/helical pitch 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 pitch.

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

61215 [Channel %1:] Block %2: Unfinished dimension incorrectly programmed

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

Definitions: Check the blank spigot dimensions. The blank spigot must be larger than the production part spigot.
Alarm triggered by following cycles: CYCLE76, CYCLE77, E_PI_CIR, E_PI_REC, E_PO_CIR, E_PO_REC, F_PI_CIR, F_PI_REC, F_PO_CIR, F_PO_REC

Remedy: Check parameters _AP1 and _AP2.

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

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

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

61230 [Channel %1:] Block %2: Tool probe diameter too small

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

Definitions: The tool probe has not been calibrated correctly.
Alarm triggered by following cycles: E_MT_CAL, E_MT_RAD, E_MT_LEN.

Remedy: Correct variable E_MESS_MT_DR[n] or E_MESS_MT_DL[n] for tool probe n+1 in data block GUD7 (measuring in JOG).

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

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

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

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

61244 [Channel %1:] Block %2: Pitch change causing

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

Definitions: The specified pitch 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 pitch 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.

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

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

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

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: pitch not possible with selected tool

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

Definitions: The pitch of the tap does not match the programmed pitch.
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 pitch.

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

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

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

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
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.

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

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

61301 [Channel %1:] Block %2: Probe not switching

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

Definitions: The measuring distance was completely traversed but no switching signal was generated at the measuring input.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: -Check measuring input.
-Check measuring distance.
-Probe defective.

61302 [Channel %1:] Block %2: Probe - collision

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

Definitions: The measuring probe collided with an obstacle when being positioned.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: ? Check spigot diameter (may be too small)
? Check measuring distance (may be to long)

61303 [Channel %1:] Block %2: Safety margin exceeded

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

Definitions: The measuring result differs greatly from the specified value.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: -Check setpoint value.
-Increase parameter _TSA.

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:

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

61306 [Channel %1:] Block %2: Permissible measuring difference exceeded

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

Definitions: Alarm triggered by following cycles: CYCLE971, CYCLE972, CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE982, CYCLE994.

Remedy: -Check setpoint value
-Increase parameter _TDIF

61307 [Channel %1:] Block %2: Incorrect measuring variant

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: The value of parameter _MVAR is impermissible.

61308 [Channel %1:] Block %2: Check measuring path _FA

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

Definitions: A traversing path for measuring was generated whose size was specified by parameter _FA that describes the maximum distance before and after the switching position (workpiece edge) and that must have a value greater than 0.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check parameter _FA.

61309 [Channel %1:] Block %2: Check probe type

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

Definitions: Probe type: 3D probe inactive.
This alarm is generated by all cycles except for CYCLE971, CYCLE972, CYCLE982.

Remedy: The probe has to be of the "3D probe" type in the tool management.
Tool type of the workpiece probe in the TO memory is impermissible.
For CYCLE971: no permissible tool probe type entered in _TP[x,8], or check the permissible working plane G17...G19 for tool type "Wheel".

61310 [Channel %1:] Block %2: Scale factor is active

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

Definitions: Scale factor = scaling is active.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Switch off the active scale factor in the program. Measuring is not possible with an active scale factor.

61311 [Channel %1:] Block %2: No D number active

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

Definitions: No tool offset for the measuring probe (for workpiece measurement) or no tool offset for the active tool (for tool measurement) is selected.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Select the tool's tool edge number D.

61312 [Channel %1:] Block %2: Check measuring cycle number

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

Definitions: The alarm can be triggered by the following measuring cycles: all measuring cycles.

Remedy: Measuring cycle called is impermissible...

61313 [Channel %1:] Block %2: Check probe number

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

Definitions: The probe has an impermissible value (_PRNUM).
The alarm can be triggered by the following measuring cycles: all measuring cycles.

Remedy: Correct _PRNUM or create data field _TP[] or _WP[] for additional tool or workpiece probe

and adjust _CVAL[0]/_CVAL[1] accordingly.

61314 [Channel %1:] Block %2: Check selected tool type

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

Definitions: Alarm is triggered: CYCLE971, CYCLE972, CYCLE982.

Remedy: Tool type impermissible for tool measurement/tool probe calibration.

61315 [Channel %1:] Block %2: Check position of cutting edge

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

Definitions: Alarm is triggered: CYCLE972, CYCLE973, CYCLE974, CYCLE982, CYCLE994.

Remedy: Check tool edge position (probe) in TO memory.

61316 [Channel %1:] Block %2: Center and radius cannot be determined

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

Definitions: No circle can be calculated from the measured points, as all measured points lie on a straight line.
The alarm is triggered by: CYCLE979

Remedy: Program change

61317 [Channel %1:] Block %2: Check parameter CYCLE116

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

Definitions: Parameterization faulty; requires 3 or 4 points to calculate the center point. Alarm is triggered: CYCLE979.

Remedy: Change parameterization of CYCLE116.

61318 [Channel %1:] Block %2: Check weighting factor _K

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

Definitions: Parameter _K is 0. Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998.

Remedy: Check parameter _K.

61319 [Channel %1:] Block %2: Check call parameter CYCLE114

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

Definitions: Internal error in measuring cycles. Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998.

Remedy: Check call parameter CYCLE114.

61320 [Channel %1:] Block %2: Check tool number

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

Definitions: With active tool management, parameter _TNUM=0 and parameter _TNAME is not set or the specified tool name of tool management is unknown.
The alarm can be triggered by the following measuring cycles: all measuring cycles.

Remedy: Check parameter _TNUM, _TNAME.

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

61322 [Channel %1:] Block %2: Check 4th number of _KNUM

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

Definitions: The specified position of _KNUM includes invalid values. Also check _MVAR.
Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998, CYCLE114.

Remedy: Check parameter _KNUM, _MVAR.

61323 [Channel %1:] Block %2: Check 5th number of _KNUM

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

Definitions: The specified position of _KNUM includes invalid values. Also check _MVAR.
Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998, CYCLE114.

Remedy: Check parameter _KNUM, _MVAR.

61324 [Channel %1:] Block %2: Check 6th number of _KNUM

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

Definitions: The specified position of _KNUM includes invalid values. Also check _MVAR.
Alarm is triggered: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994, CYCLE998, CYCLE114.

Remedy: Check parameter _KNUM, _MVAR.

61325 [Channel %1:] Block %2: Check measuring axis/auxiliary measuring axis

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

Definitions: Parameter for metering shaft _MA has an incorrect value.
Alarm is triggered by: all measuring cycles except for CYCLE979.

Remedy: Check parameter _MA.

61326 [Channel %1:] Block %2: Check measuring direction

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

Definitions: Parameter for measuring direction _MD has an incorrect value. Alarm is triggered: CYCLE973, CYCLE976.

Remedy: Check parameter _MD.

61327 [Channel %1:] Block %2: Program reset required

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

Definitions: NC reset required.
Alarm is triggered: all measuring cycles except for CYCLE973, CYCLE976.

Remedy: Execute NC reset.

61328 [Channel %1:] Block %2: Check D number

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

Definitions: D number in parameter _KNUM is 0.
The alarm can be triggered by all measuring cycles.

Remedy: Check parameter _KNUM.

61329 [Channel %1:] Block %2: Check rotary axis

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

Definitions: No name assigned to the axis number specified under parameter _RA or axis not configured as rotary axis. Alarm is triggered: CYCLE998.

Remedy: Check MD 20080 or MD 30300.

61330 [Channel %1:] Block %2: Coordinate rotation active

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

Definitions: No measuring possible in the rotated coordinate system. Alarm is triggered: CYCLE972, CYCLE973, CYCLE974, CYCLE994.

Remedy: Check the conditions for measuring.

61331 [Channel %1:] Block %2: Angle too large, change measuring axis

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

Definitions: Parameter _STA is too large of the specified metering shaft. Alarm is triggered: CYCLE998.

Remedy: Select another metering axis.

61332 [Channel %1:] Block %2: Modify tool tip position

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

Definitions: The tool tip is below the measuring probe surface (e.g. for a ring gauge or cube). Alarm is triggered: CYCLE971, CYLCE972, CYCLE982, E_MT_CAL, E_MT_LEN, E_MT_RAD.

Remedy: Place the tool above the measuring probe surface.

61333 [Channel %1:] Block %2: Check calibration facility number

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

Definitions: Parameter _CALNUM is too large. Alarm is triggered: CYCLE973.

Remedy: Reduce _CALNUM to a permissible value or increase maximum value _CVAL[2] in GUD6.

61334 [Channel %1:] Block %2: Check safety area

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

Definitions: Parameter _SZA, _SZO too large or too small. Alarm is triggered: CYCLE977.

Remedy: Check parameter _SZA, _SZO.

61336 [Channel %1:] Block %2: Geometry axes do not exist

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

Definitions: No geometry axes configured. Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Machine data in MD 20060 must be changed.

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

61338 [Channel %1:] Block %2: Positioning speed equal to zero

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

Definitions: For some measuring versions, for example measuring spigots, in addition to the actual measuring paths, intermediate paths are generated that are traversed with a specified feed. The values for the feed are specified in parameters _SPEED[1] and _SPEED[2] in GUD6.
Alarm triggered by following measuring cycles: all measuring cycles.

Remedy: Check parameter _SPEED[1], _SPEED[2] in GUD6.

61339 [Channel %1:] Block %2:Correction factor for rapid traverse speed = 0

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check parameter _SPEED[0] in GUD6.

61340 [Channel %1:] Block %2: Incorrect alarm number

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Internal error in measuring cycles.

61341 [Channel %1:] Block %2: Probe not calibrated in active plane.

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

Definitions: The alarm is triggerd by the following cycles: CYCLE974, CYCLE977, CYCLE978, CYCLE979.

Remedy: Calibrate the probe prior to calling a cycle.

61342 [Channel %1:] Block %2: Invalid software version or wrong format entered in GUD6

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Lower than measuring cycle software 6.2:_SI[1] in GUD6 has no value or a value < 3
Higher than measuring cycle software 6.3: Upgrade NCK software version.

61343 [Channel %1:] Block %2: No tool available for specified tool identifier

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check name of tool identifier.

61344 [Channel %1:] Block %2: Several tools are active

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Remove tool from another spindle.

61345 [Channel %1:] Block %2: Parameterized D number (_KNUM) too large

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Reduce the D number in _KNUM, check software or MD of flat D number.

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

61347 [Channel %1:] Block %2: Angle 1st edge - 2nd edge equals 0

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

Definitions: The alarm is triggerd by the following cycles: CYCLE961.

Remedy: Parameter _INCA equals 0.

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

61349 [Channel %1:] Block %2: Distance upper probe edge - measuring position = 0 for tool radius measurement

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

Definitions: The alarm is triggerd by the following cycles: CYCLE971.
Parameter _TP[x,9] distance between upper edge and lower edge of tool probe equals 0; relevant for radius measurement.

Remedy: Check parameter _TP[x,9].

61350 [Channel %1:] Block %2: feed, speed not programmed in _MFS for tool measurement with rotating spindle

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

Definitions: The alarm is triggerd by the following cycles: CYCLE971.
Measuring feed and/or spindle speed of tool measurement with rotating spindle in GUD variable _MFS[2] not specified.

Remedy: Check parameter _MFS[2].

61351 [Channel %1:] Block %2: Tool length or radius is 0

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

Definitions: The alarm is triggerd by the following cycles: CYCLE971.
For the active tool, the length or radius equal zero.

Remedy: Check length and radius of the active tool in the compensation data memory.

61352 [Channel %1:] Block %2: Path for logfile not permitted

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
The specified path for the log file is incorrect.

Remedy: Check parameter _PROTNAME[1].

61353 [Channel %1:] Block %2: Path for logfile not found

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
The specified directory does not exist or the specified path is incorrect.

Remedy: Check parameter _PROTNAME[1].

61354 [Channel %1:] Block %2: File for logfile not found

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
No name specified for the log file.

Remedy: Check parameter _PROTNAME[1].

61355 [Channel %1:] Block %2: Incorrect file type for logfile

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
The file extension for the log file is incorrect.

Remedy: Check parameter _PROTNAME[1].

61356 [Channel %1:] Block %2: File for logfile is being used

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
The log file is already used by an NC program.

Remedy: Check parameter _PROTNAME[1].

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

61358 [Channel %1:] Block %2: Error during recording

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
Internal error

Remedy: Call the hotline!

61359 [Channel %1:] Block %2: - continue with RESET

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

Definitions: The alarm is triggerd by the following cycles: CYCLE106.
Internal error

Remedy: Call the hotline!

61360 [Channel %1:] Block %2: Log job undefined - continue with RESET

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

Definitions: Alarm triggered by following cycle: CYCLE106.
Cycle CYCLE106 was called by an incorrect parameter.

Remedy: Check cycle call for CYCLE106, specifically the call parameter.

61361 [Channel %1:] Block %2: Variable cannot be recorded

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

Definitions: The alarm is triggerd by the following cycles: CYCLE105.
The value specified in _PROTVAL[] cannot be logged.

Remedy: Check parameter _PROTVAL[].

61362 [Channel %1:] Block %2: Cycle118: No. of values too large

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

Definitions: The alarm is triggerd by the following cycles: CYCLE118.
4th parameter for CYCLE118 is larger than 10.

Remedy: Reduce the 4th parameter (PAR4) of CYCLE118.

61363 [Channel %1:] Block %2: Max. no. of value lines for recording exceeded

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

Definitions: Maximum number of value lines exceeded.
Alarm triggerd by following cycle: CYCLE105.

Remedy: Reduce the number of value lines.
Check parameter _PROTFORM[4].

61364 [Channel %1:] Block %2: Check distance from measuring point 1 to measuring point2

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

Definitions: The alarm is triggerd by the following cycles: CYCLE998.
Parameter _ID is <= 0.

Remedy: Check parameter _ID.

61365 [Channel %1:] Block %2: Check circular feed

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

Definitions: The alarm is triggerd by the following cycles: CYCLE979.
Parameter _RF is <= 0.

Remedy: Check parameter _RF.

61366 [Channel %1:] Block %2: Direction of rotation for tool measurement with rotating spindle not specified in _CM[5]

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

Definitions: The alarm is triggerd by the following cycles: CYCLE971.
Permissible values for data field _CM[5] in GUD6 block are 3 (corresponds to M3) or 4 (corresponds to M4).

Remedy: Check parameter _CM[5] in GUD6.

61367 [Channel %1:] Block %2: Parameters _SETV[0...3] or _SETV[4...7] are identical

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

Definitions: The alarm is triggerd by the following cycles: CYCLE961.

Remedy: Specify different positions for the relevant points of _SETV[0...7] .

61368 [Channel %1:] Block %2: Straights through parameter _SETV[0...3] or _SETV[4...7]do not intersect

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

Definitions: The alarm is triggerd by the following cycles: CYCLE961.

Remedy: Specify different positions for the relevant points of _SETV[0...7] .

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

61370 [Channel %1:] Block %2: _PROTVAL[0] -_PROTVAL[5] do not have any entries

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

Definitions: The alarm is triggerd by the following cycles: CYCLE105.

Remedy: Enter values in _PROTVAL[0...5].

61371 [Channel %1:] Block %2: Product of column width and number of columns exceeds 200 characters per line

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

Definitions: The alarm is triggerd by the following cycles: CYCLE105.

Remedy: Reduce the column width (_PROTFORM[4]) or number of columns (_PROTVAL[2...5]).

61372 [Channel %1:] Block %2: selected meas.variant requires SPOS-capable spindle

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Change measuring variant or check machine equipment.

61373 [Channel %1:] Block %2: Mono-directional probe requires SPOS-capable spindle

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check machine equipment.

61401 [Channel %1:] Block %2: Probe not switching, traversing path limitation through software limit position

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

Definitions: The alarm is triggerd by the following cycles: CYCLE961, CYCLE971, CYCLE976, CYCLE977, CYCLE978, CYCLE998
The position defined by a setpoint value cannot be reached, as the software limit position has been exceeded.

Remedy: Check specified setpoint value.

61402 [Channel %1:] Block %2: Probe collision, traversing path limitation through software limit position

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

Definitions: The alarm is triggerd by the following cycles: CYCLE977
For the measuring variant Measure web/shaft, the position path in the plane was limited by the software limit position. The probe switched in the following infeed along the infeed axis.

Remedy: Check programmed position regarding software end position.

61403 [Channel %1:] Block %2: Internal cycle error during frame calculation.

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Call the SIEMENS hotline

61404 [Channel %1:] Block %2: Internal cycle error during tool offset.

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check the dependent tool specifications.

61405 [Channel %1:] Block %2: tool environment does not exist in _TENV

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

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Correct the names or create this environment.

61406 [Channel %1:] Block %2: check DL number in _DLNUM

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check the number of additive offset and setup offset.
Check parameter _DLNUM.

61407 [Channel %1:] Block %2: check 7th digit and higher of _KNUM

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The 6th position of _KNUM includes invalid values.
Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Check the number of additive offset and setup offset.
Check parameter _KNUM.

61408 [Channel %1:] Block %2: total offsets not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Set MD 18080, Bit 8=1

61409 [Channel %1:] Block %2: set up offsets not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: Set MD 18112, Bit 4=1

61410 [Channel %1:] Block %2: option or offset value not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: The variable to be corrected requires an option or an increase in MD values.

61411 [Channel %1:] Block %2: frame calculation impossible, check values

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE997, CYCLE119

Remedy: Check the setpoint and actual values

61412 [Channel %1:] Block %2: channel basic frame not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE997, CYCLE119
Remedy: Set MD 28081>0, $P_CHBFRMASK>0

61413 [Channel %1:] Block %2: check setpoint of ball diameter, _SETVAL<=0

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm is triggerd by the following cycles: CYCLE997
Remedy: Check setpoint value of spherical diameter.

61414 [Channel %1:] Block %2 : distortion of triangle over limit

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm is triggerd by the following cycles: CYCLE997, CYCLE119
Remedy: Check the setpoint and actual values

61415 [Channel %1:] Block %2: check probe / machining plane

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm is triggerd by the following cycles: CYCLE971
Remedy: Enter permissible probe (_TP[x,8], _TPW[x,8]) for machining plane or change machining plane.

61416 [Channel %1:] Block %2: adapt array size %4!

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: "Adjust field size _TP[ ]/_CVAL[0]!" or "Adjust field size _WP[ ]/_CVAL[1]!" or "Adjust field size _KP[ ]/_CVAL[2]!" or "Adjust field size _TWP[ ]/_CVAL[3]!". Check probe/machining plane.
Alarm triggered by following measuring cycles: all measuring cycles.
Remedy: Adjust _CVAL entry with the number of available probe or calibration block data fields.

61417 [Channel %1:] Block %2: Probe will collide with the carrier of the reference groove.

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following measuring cycles: CYCLE973
Remedy: Take up collision-free initial position of the axes involved in the measuring process.

61418 [Channel %1:] Block %2: Protocol file too small, check MD11420: LEN_PROTOCOL_FILE.

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: Alarm can be triggered by following measuring cycle: CYCLE106
Remedy: Check MD11420: LEN_PROTOCOL_FILE.

61419 [Channel %1:] Block %2: Check probe calibration with reference to center of ball/circumference of ball.

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following measuring cycles: CYCLE974, CYCLE994, CYCLE977, CYCLE978, CYCLE979, CYCLE997, CYCLE998
Remedy: The workpiece probe must be calibrated according to its use in the measuring cycles.

61420 [Channel %1:] Block %2: Check calibration of multi/mono probes.

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following measuring cycles: CYCLE974, CYCLE994, CYCLE977, CYCLE978, CYCLE979, CYCLE997, CYCLE998
Remedy: The workpiece probe must be calibrated according to its type and use.

61421 [Channel %1:] Block %2: Software release of measuring cycles or NCK inadequate or set up incorrectly --> error code %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> _OVR[] - parameter field too small. Check GUD definition.
DEF CHAN REAL _OVR[72] (up to MZ06.03.xx.xx =32)

61422 [Channel %1:] Block %2: Parameter _MVAR incorrect --> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> _MVAR = 9x identifier CYCLE996 measure kinematics
2. Error code = B -> Parameter for normalizing (_MVAR) incorrect
3. Error code = C -> Measurement variant "compute only" active, but rotary axis 1 or 2 not measured
(see also parameter _OVR[40])

61423 [Channel %1:] Block %2: Parameter _TNUM not agreed or not created

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Parameter CYCLE996 _TNUM incorrect or equals zero
2. No swivel data record created -> MD18088 = 0

61424 [Channel %1:] Block %2: Parameter _SETVAL for diameter of calibration ball incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Check parameter _SETVAL.

61425 [Channel %1:] Block %2: Parameter for measuring axis rotary axis 1 or 2 incorrect -> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> Rotary axis number incorrect (1 or 2)
2. Error code = B -> No name agreed for rotary axis 1
3. Error code = C -> Rotary axis vector 1 equals zero
4. Error code = D -> No name agreed for rotary axis 2
5. Error code = E -> Rotary axis vector 2 equals zero

61426 [Channel %1:] Block %2: Sum of the active offsets does not equal zero -> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
Check the overview of active offsets ($P_ACTFRAME)
1. Error code = A -> Sum of the translatory offsets of the geometry axes <> 0
2. Error code = B -> Sum of the fine offsets of the geometry axes <> 0
3. Error code = C -> Sum of the rotary components of the geometry axes <> 0
4. Error code = D -> Sum of the translatory offsets of rotary axis 1 <> 0
5. Error code = E -> Sum of the translatory offsets of rotary axis 2 <> 0

61427 [Channel %1:] Block %2: Tool data of the active workpiece probe incorrect or inactive --> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> Workpiece probe (or tool edge) inactive
2. Error code = B -> Length L1 of the workpiece probe = 0

61428 [Channel %1:] Block %2: Error while creating log file -> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> Number of log files in the current directory > 99
2. Error code = B -> Log files too long. Rename or delete log files,
check MD11420 $MN_LEN_PROTOCOL_FILE.

61429 [Channel %1:] Block %2: Measuring axis (rotary axis 1 or 2) not in basic or intended position -> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> Rotary axis 1 not in basic position on 1st measurement
2. Error code = B -> Rotary axis 2 not in basic position on 1st measurement
3. Error code = C -> Rotary axis 2 not in intended position on 2nd or 3rd measurement in comparison to 1st measurement, see parameters _OVR[63 to 65]
4. Error code = D -> Rotary axis 1 not in intended position on 2nd or 3rd measurement in comparison to 1st measurement, see parameters _OVR[60 to 62]

61430 [Channel %1:] Block %2: Kinematic vectors not computed -> Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE996

Remedy: Causes of error:
1. Error code = A -> Plausibility of the input points PM1, PM2, PM3 not fulfilled, resulting side lengths must be not equal to zero
(Notice: even in the case of side lengths not equal to zero, there is a risk of not being able to form a triangle => check PM1...3.)
2. Error code = B -> Enclosed angle at PM1 between the spread vectors PM1PM2 and PM1PM3 is equal to zero 0.
Starting points do not form a triangle.
3. Error code = C -> Enclosed angle at PM2 between the spread vectors PM2PM1 and PM2PM3 is equal to 0.
Starting points do not form a triangle.
4. Error code = D -> Enclosed angle at PM3 between the spread vectors PM3PM1 and PM3PM2 is equal to 0.
Starting points do not form a triangle.
5. Error code = E -> Normalizing interpolation point: Invalid axis name defined for computation
6. Error code = F -> Normalizing vertice: Invalid plane defined for computation

61440 [Channel %1:] Block %2: Position of cutting edge cannot be determined

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycle: CYCLE982

Remedy: A turning tool with a cutting edge position between 1 and 8 must be used as the tool type.
Check the entered cutting edge position with reference to the basic position of the tool carrier.

61441 [Channel %1:] Block %2: Position of cutting edge is not in the machining plane.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycle: CYCLE982

Remedy: The position of the cutting edge of the turning tool (cutting tip) is no longer in the machining plane (interpolation plane), this can be caused, for example, by a tool carrier with orientation capability. Correct the tool carrier position.

61442 [Channel %1:] Block %2: Tool carrier not parallel to the geometry axes

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycle: CYCLE982

Remedy: As a result of incorrect positioning of the orientable tool carrier, tool lengths L1,2,3 are not parallel to the geometry axes.
Optimize the axes of the tool carrier.

61443 [Channel %1:] Block %2: Advance angle _INCA=0 or greater/less than +/-90° or +/-120°

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycle: CYCLE979

Remedy: Check the value in the parameter advance angle _INCA.
If 3-point measurement is selected, _INCA must not be greater/less than +/-120°, and with 4-point measurement _INCA must not be greater/less than +/-90°.
The advance angle _INCA must always be parameterized unequal to "zero".

61444 [Channel %1:] Block %2: Current measuring speed is not identical to the calibration speed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm can be triggered by the following measuring cycles: CYCLE974, CYCLE994, CYCLE977, CYCLE978, CYCLE979, CYCLE997, CYCLE998
E_MS_CAN, E_MS_HOL, E_MS_POC, E_MS_PIN, E_MS_SPI

Remedy: 1. Repeat the calibration on the basis of the desired measuring speed.
2. Match the current measuring speed to the calibration speed.
Note: In each case, the relevant calibration speed is stored in each calibration data record.

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

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

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

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

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

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

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
Remedy: Check values in grinding wheel data
Program Continuation: Clear alarm with the RESET key. Restart part program

61560 [Channel %1:] Block %2: Infeed in Z direction too big per stroke, or wheel too narrow

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE427, CYCLE428
Remedy: Reduce infeed path parameter or use other tool
Program Continuation: Clear alarm with the RESET key. Restart part program

61561 [Channel %1:] Block %2: Feed left wheel edge <=0

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

61562 [Channel %1:] Block %2: Feed right wheel edge <=0

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

61563 [Channel %1:] Block %2: Feed on the diameter <=0

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

61564 [Channel %1:] Block %2: Feed insertion <=0

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE434, CYCLE444
Remedy: Check values in grinding wheel data
Program Continuation: Clear alarm with the RESET key. Restart part program

61565 [Channel %1:] Block %2: Feed dressing <=0

Parameters: %1 = Channel number
%2 = Block number, label channel number
Definitions: The alarm can be triggered by the following grinding cycles: CYCLE434, CYCLE444

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: %1 = Channel number
%2 = Block number, label
Definitions: The programmed radius of the machined part is too small. Alarm triggered by following cycles: CYCLE94, CYCLE96.
Remedy: Check parameter SPD or DIATH.
Program Continuation: Clear alarm with the RESET key. Restart part program

61602 [Channel %1:] Block %2: Tool width incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Plunge cutter is larger than the programmed groove width. Alarm triggered by following cycle: CYCLE93.
Remedy: Check tool or change program.
Program Continuation: Clear alarm with the RESET key. Restart part program

61603 [Channel %1:] Block %2: Recess type incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Radii/chamfers at the groove base do not match the groove width. Face groove on a contour element running parallel to the longitudinal axis is not possible. Alarm triggered by following cycle: CYCLE93.
Remedy: Check parameter VARI.
Program Continuation: Clear alarm with the RESET key. Restart part program

61604 [Channel %1:] Block %2: Active tool violates programmed contour

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Contour violation in the relief cut elements due to the tool clearance angle of the tool used. Alarm triggered by following cycle: CYCLE95.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use a different tool or check the contour subroutine.
Program Continuation: Clear alarm with the RESET key. Restart part program

61605 [Channel %1:] Block %2: Contour incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Illegal relief cut element detected. Alarm triggered by following cycles: CYCLE76, CYCLE77, CYCLE95.
Remedy: Check contour program.
Program Continuation: Clear alarm with the RESET key. Restart part program

61606 [Channel %1:] Block %2: Error during contour preparation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: An error has been found on conditioning the contour. This alarm is always related to one of NCK alarms 10930...10934, 15800 or 15810. Alarm triggered by following cycle: CYCLE95.
Remedy: Check contour subroutine.

Program Continuation: Clear alarm with the RESET key. Restart part program

61607 [Channel %1:] Block %2: Starting point incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The starting point reached before the cycle call does not lie outside the rectangle described by the contour subroutine. Alarm triggered by following cycle: CYCLE95.

Remedy: Check starting point prior to cycle call.

Program Continuation: Clear alarm with the RESET key. Restart part program

61608 [Channel %1:] Block %2: Incorrect tool point direction programmed

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: 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: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE94, CYCLE96, LONGHOLE, POCKET3, SLOT1.

Remedy: Check parameter for the undercut form or groove form or pocket.

Program Continuation: Clear alarm with the RESET key. Restart part program

61610 [Channel %1:] Block %2: No infeed depth programmed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE76, CYCLE77, CYCLE96.

Remedy: Check parameter MID.

Program Continuation: Clear alarm with the RESET key. Restart part program

61611 [Channel %1:] Block %2: No point of intersection found

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No intersection could be calculated with the contour. Alarm triggered by following cycle: CYCLE95.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check contour programming or modify infeed depth.

Program Continuation: Clear alarm with the RESET key. Restart part program

61612 [Channel %1:] Block %2: Thread finishing not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE97, CYCLE98.

Remedy: Check the conditions for thread finishing.

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

61701 [Channel %1:] Block %2: Error in finished part contour description

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Either none of parameters _NP1, _NP2 and _NP3 supplied or error in finished part contour programming.
Alarm triggered by following cycle: CYCLE950

Remedy: - Check parameters _NP1, _NP2 and _NP3.
- Check finished-part contour programming.

61702 [Channel %1:] Block %2: Error in blank contour description

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Either none of parameters _NP5, _NP6 and _NP7 supplied or error in blank contour programming.
The alarm is triggered by the following cycles: CYCLE950

Remedy: - Check parameters _NP5, _NP6 and _NP7.
- Check blank contour programming.

61703 [Channel %1:] Block %2: Internal cycle error while deleting file

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61704 [Channel %1:] Block %2: Internal cycle error while writing to file

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61705 [Channel %1:] Block %2: Internal cycle error while reading to file

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61706 [Channel %1:] Block %2: Internal cycle error while generating checksum

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61707 [Channel %1:] Block %2: internal cycle error with ACTIVATE at HMI

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61708 [Channel %1:] Block %2: internal cycle error with READYPROG at HMI

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE950, CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61709 [Channel %1:] Block %2: Timeout in contour calculation

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

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

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

61720 [Channel %1:] Block %2: Incorrect parameter input.

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

61721 [Channel %1:] Block %2: Error contour direction cannot be determined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

61722 [Channel %1:] Block %2: System error

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61723 [Channel %1:] Block %2: Machining not possible

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: Use a tool with a larger clearance angle.

61724 [Channel %1:] Block %2: Material not available

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61725 [Channel %1:] Block %2: Memory space problem, therefore error in contour generating

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61726 [Channel %1:] Block %2: Internal error: Memory space problem _FILECTRL_INTERNAL_ERROR

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61727 [Channel %1:] Block %2: Internal error: Memory space problem _FILECTRL_EXTERNAL_ERROR

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61728 [Channel %1:] Block %2: Internal error: Memory space problem _ALLOC_P_INTERNAL_ERROR

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61729 [Channel %1:] Block %2: Internal error: Memory space problem _ALLOC_P_EXTERNAL_ERROR

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61730 [Channel %1:] Block %2: Internal error: invalid memory

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

61731 [Channel %1:] Block %2: Internal error: floating point exception

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61732 [Channel %1:] Block %2: Internal error: invalid instruction

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61733 [Channel %1:] Block %2: Internal error: Floating_Point_Error

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61734 [Channel %1:] Block %2: Cutting edge not compatible with cutting direction

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61735 [Channel %1:] Block %2: Finished part not within blank contour

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: Check blank contour definition.

61736 [Channel %1:] Block %2: Insert length of tool < machining depth

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61737 [Channel %1:] Block %2: Machining_cutting_depth > tool nose radius

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61738 [Channel %1:] Block %2: Machining_cutting_depth < tool nose radius

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61739 [Channel %1:] Block %2: Wrong insert position of tool for this machining operation

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycle: CYCLE950.
Remedy: --

61740 [Channel %1:] Block %2: Blank must be closed contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: Check whether blank contour is closed, i.e. starting point = end point.

61741 [Channel %1:] Block %2: Cancel due to insufficient memory space

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

61742 [Channel %1:] Block %2: Approach collision, correction not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE950.

Remedy: --

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: --

Program Continuation: Clear alarm with the RESET key. Restart part program

61800 [Channel %1:] Block %2: Ext. CNC system missing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Machine data for external language MD18800: $MN_MM_EXTERN_LANGUAGE or option bit 19800 $ON_EXTERN_LANGUAGE is not set.

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

61801 [Channel %1:] Block %2: Wrong G code selected

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the program call CYCLE300<value> an impermissible numerical value was programmed for the entered CNC System, or in the Cycles Setting Datum an incorrect value for the G Code System was set.

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

61802 [Channel %1:] Block %2: Wrong axis type

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed axis is assigned to a spindle

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

61803 [Channel %1:] Block %2: Programmed axis not available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed axis is not in the system.
Alarm triggered by following cycles: CYCLE83, CYCLE84, CYCLE840.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check parameter _AXN.
Check MD20050-20080.

Program Continuation: Clear alarm with the RESET key. Restart part program

61804 [Channel %1:] Block %2: Progr. position exceeds reference point

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed intermediate position or actual position is behind the reference point.

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

61805 [Channel %1:] Block %2: Value programmed absolute and incremental

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The programmed intermediate position is both absolutely as well as incrementally programmed.

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

61806 [Channel %1:] Block %2: Wrong axis assignment

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The axis-assignment sequence is wrong.

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

61807 [Channel %1:] Block %2: Wrong spindle direction programmed (active)

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE840.
The programmed spindle direction contradicts the spindle direction planned for the cycle.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check parameters SDR and SDAC.

Program Continuation: Clear alarm with the RESET key. Restart part program

61808 [Channel %1:] Block %2: Final drilling depth or single drilling depth missing

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The total depth Z or individual drilling depth Q is missing from theG8xblock (initial cycle call).

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

61809 [Channel %1:] Block %2: Drill position not permissible

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

61810 [Channel %1:] Block %2: ISO G code not possible

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the call block an impermissible ISO axis name was programmed.

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: %1 = Channel number
%2 = Block number, label

Definitions: In the call block an impermissible numerical value was programmed.

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

61812 [Channel %1:] Block %2: Value(s) in external cycle call wrongly defined

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: In the call block an impermissible numerical value was programmed.

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

61813 [Channel %1:] Block %2: GUD value wrongly defined

Definitions: An impermissible numerical value was
entered in the cycles-setting data.

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

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: %1 = Channel number
%2 = Block number

Definitions: G40 was inactive before the cycle call.

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

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

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

61900 [Channel %1:] Block %2: No contour available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61901 [Channel %1:] Block %2: Contour not closed

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61902 [Channel %1:] Block %2: No more memory available

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61903 [Channel %1:] Block %2: Too many contour elements

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61904 [Channel %1:] Block %2: Too many intersections

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61905 [Channel %1:] Block %2: Cutter radius too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The diameter of the cutter used is too small, residual material is left in the slot. Alarm triggered by following cycles: SLOT2, CYCLE73, CYCLE74, CYCLE75.

Remedy: Use a tool with a larger radius.

61906 [Channel %1:] Block %2: Too many contours

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61907 [Channel %1:] Block %2: No center point specified for circle

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61908 [Channel %1:] Block %2: No starting point specified

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61909 [Channel %1:] Block %2: Helix radius too small

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61910 [Channel %1:] Block %2: Helix violates contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61911 [Channel %1:] Block %2: Several approach points required

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

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

61913 [Channel %1:] Block %2: No residual material generated

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61914 [Channel %1:] Block %2: Programmed helix violates contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61915 [Channel %1:] Block %2: Approach/retract motion violates contour

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61916 [Channel %1:] Block %2: Ramp path too short

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61917 [Channel %1:] Block %2: Residual corners may remain with overlapping of less than 50%

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

61918 [Channel %1:] Block %2: Cutter radius for residual material too large

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61980 [Channel %1:] Block %2: Error in the island contour

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61981 [Channel %1:] Block %2: Error in the edge contour

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61982 [Channel %1:] Block %2: Infeed width in the plane too large

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61983 [Channel %1:] Block %2: Pocket edge contour missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61984 [Channel %1:] Block %2: Tool parameter _TN not defined

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61985 [Channel %1:] Block %2: Program name for drilling positions missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61986 [Channel %1:] Block %2: Program pocket milling missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61987 [Channel %1:] Block %2: Program drilling positions missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61988 [Channel %1:] Block %2: Program name for pocket milling missing

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.
Remedy: --

61989 [Channel %1:] Block %2: D1 is not programmed as active tool cutting edge

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycles: CYCLE73, CYCLE74, CYCLE75.

Remedy: --

62000 [Channel %1:] Block %2: Insert new tool

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Please load new tool.

Remedy: --

Program Continuation: Clear alarm with the Delete key or NC START.

62100 [Channel %1:] Block %2: No drilling cycle active

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: No modal drilling cycle has been called before the drilling pattern cycle call. Alarm triggered by following cycles: HOLES1, HOLES2.

Remedy: Check whether a drilling cycle was called prior to calling the drilling pattern cycle.

Program Continuation: Clear alarm with the Delete key or NC START.

62101 [Channel %1:] Block %2: Milling direction incorrect - G3 is generated

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Synchronous or reverse rotation programmed. But the spindle does not rotate at a cycle call.

Remedy: Check value in paramter CDIR.

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.

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.

62105 [Channel %1:] Block %2: Number of columns or lines equals zero

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE801.

Remedy: Check parameters _NUM1 and _NUM2.

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.

62180 [Channel %1:] Block %2: Set rotary axes %4 [deg]

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE800.
Note on 62180 and 62181:
Sample display of the swivel angle to be set for a manual rotary axis in CYCLE800:
62181 "Set rotary axis B: 32.5 [grd]"

Remedy: Settable angles for manual rotary axes.

62181 [Channel %1:] Block %2: Set rotary axis %4 [deg]

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: Alarm triggered by following cycle: CYCLE800.
Note on 62180 and 62181:
Sample display of the swivel angle to be set for a manual rotary axis in CYCLE800:
62181 "Set rotary axis B: 32.5 [grd]"

Remedy: Settable angle for manual rotary axis.

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: --
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.

62185 [Channel %1:] Block %2 : angle adapted to angle grid: %4

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: %4 difference angle with Hirth tooth system
Alarm triggered by following cycle: CYCLE800.
Remedy: Check installation and start-up of the swivel cycle CYCLE800.

62186 [Channel %1:] Block %2: 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 cycle: CYCLE800.
Note for 62186 and 62187:
Set and active error messages 62186 and 62187 with GUD7 parameter _TC_FR:
100th position 0xx -> no error analysis 62186 61287
1xx -> error analysis 62186 - active WO G%4 and basic frames include rotations
2xx -> error analysis 62187 - several active basic frames (G500) include rotations
3xx -> error analyses 62186 and 62187.
Remedy: For %4 of the active WO see notes for 62186 and 62187.

62187 [Channel %1:] Block %2: Swiveling in JOG --> several active basic frames (G500) contain rotations

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Several active basic frames (G500) include rotations.
Alarm triggered by following cycle: CYCLE800.
Note on 62186 and 62187
Setting of the activation of error messages 62186 and 62187 with GUD7 parameter _TC_FR:
100's place 0xx -> no error evaluation 62186 61287
1xx -> error evaluation 62186 - active NV G%4 and basic frames include rotations
2xx -> error evaluation 62187 - several active basic frames (G500) include rotations
3xx -> error evaluation 62186 and 62187
Remedy: See notes for 62186 and 62187.

62200 [Channel %1:] Block %2: Start spindle

Parameters: %1 = Channel number
%2 = Block number, label
Definitions: Stop prior to thread machining, as the spindle is in stop position.

Alarm triggered by following cycles: ASUP, E_TR_CON, F_TR_CON.

Remedy: Start the tool spindle before machining the thread.

62201 [Channel %1:] Block %2: Z offset does not influence the retraction planes.

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: The retraction planes refer to the workpiece. Therefore, programmable offsets do not influence the retraction planes.
Alarm triggered by following cycle: F_SP_RP.

Remedy: Ensure that the offset will not cause a collision.
Then start the NC.
The alarm can be suppressed via display machine data 9898.

62202 [Channel %1:] Block %2: NOTICE: tool travels directly to machining!

Parameters: %1 = Channel number
%2 = Block number, label

Definitions: After block search a position is to be reached by direct approach.
Alarm triggered by following cycle: F_TFS.

Remedy: Check whether the desired position can be reached without collision.
Then execute an 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.

62303 [Channel %1:] Block %2: Safety margin exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm can be triggered by following measuring cycles: all measuring cycles.

Remedy: -Check setpoint value
-Increase parameter _TSA

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.

62305 [Channel %1:] Block %2: Dimension too small

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE994

Remedy: The difference between actual and setpoint value is smaller than lower tolerance limit (parameter _TLL).

62306 [Channel %1:] Block %2: Permissible measuring difference exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE971, CYCLE972, CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE982, CYCLE994

Remedy: The difference between actual and setpoint value is larger than tolerance parameter _TDIF, tool data are not corrected.

62307 [Channel %1:] Block %2: Maximum number of characters per line exceeded.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggerd by the following cycles: CYCLE105
Insufficient number of characters per line.

Remedy: Increase the value in _PROTFORM[1]

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.

62310 [Channel %1:] Block %2: The max. number of characters per line is limited to 200 characters per line

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycles: CYCLE105.
The maximum number of characters per line has been limited to 200 characters per line.

Remedy: --

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

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.

62313 **[Channel %1:] Block %2: The number of lines per page _PROTFORM[0] is incorrect and is automatically adjusted.**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: Alarm triggered by following cycle: CYCLE106.

Remedy: Check _PROTFORM[0] in the program.

62314 **[Channel %1:] Block %2: Traverse path limitation via software end position, collision detection activated, continue with NC START / cancel with RESET.**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggered by the following cycle: CYCLE977

Remedy: Position the workpiece to be measured further away from the software end positions.

62315 **[Channel %1:] Block %2: Overwrite swivel data record TCARR = %4, yes -> NC start, no -> reset**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggered by the following cycle: CYCLE996

Remedy:

62316 **[Channel %1:] Block %2: Overwrite TRAORIdata, yes -> NC start, no -> reset**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggered by the following cycle: CYCLE996

Remedy:

62317 **[Channel %1:] Block %2: Tolerance of the linear vector %4 exceeded**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggered by the following cycle: CYCLE996

Remedy:

62318 **[Channel %1:] Block %2: Tolerance of the rotary axis vector %4 exceeded**

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions: The alarm is triggered by the following cycle: CYCLE996

Remedy:

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.

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

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: --

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: --

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.

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

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.

62934 [Channel %1:] Block %2: Incorrect finishing contour calculation

Parameters: %1 = Channel number
%2 = Block number, label channel number

Definitions:

Remedy: --

2.3 Drive alarms

300406 Problem in the non-cyclic communication for basic address %1, additional information %2, %3, %4

Definitions: For PROFIdrive only:
A problem occured during the non-cyclic communication with the logical start address. The additional information defines the location of the problem. If the logical start address 0 is output, only the additional information is relevant.

Reaction: Alarm display.
Warning display.

Remedy: Please inform the authorized personnel/service department. The alarm can be suppressed with MD11411 $MN_ENABLE_ALARM_MASK bit 1 == 0
SIEMENS AG, System Support for A&D MC products, Hotline (Phone: see alarm 1000)

Program Continuation: Clear alarm with the Delete key or NC START.

300410 Axis %1 drive %2 error when storing a file (%3, %4)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Definitions: An attempt to save a data block, e. g. the result of a measuring function, in the file system has failed.
On error code 1 == 291: An error occurred during preparation of the ACC information. Basic information prepared on the drive contains an error or has an unknown format.
On error code 1 == 292: Memory shortage during preparation of the ACC information.

Reaction: Interface signals are set.
Alarm display.

Remedy: - Please inform the authorized personnel/service department.
- Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself (this will require a complete data backup).
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of
- 18270 $MM_NUM_SUBDIR_PER_DIR,
- 18310 $MM_NUM_DIR_IN_FILESYSTEM,
- Power On
- Reload saved data
- On error code 1 == 291: Replace the drive software and use version with suitable ACC basic information.
- On error code 1 == 292: Replace the drive software and use fewer different versions of the drive software.

Program Continuation: Clear alarm with the RESET key. Restart part program

300411 Axis %1 drive %2 error when reading a file (%3, %4)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Definitions: An attempt to read a data block, e.g. a drive boot file, from the file system has failed. The data block or the file system is damaged.

Reaction: Interface signals are set.

Alarm display.

Remedy: If the error occurred during power-up, i.e. it is probably connected to a drive boot file, delete all boot files and load them back into the control from the back-up copy.

Program Continuation: Clear alarm with the RESET key. Restart part program

300412 Error when storing a file (%1, %2)

Parameters: %1 = Error code 1
%2 = Error code 2

Definitions: An attempt to save a data block, e.g. the result of a measuring function, in the file system has failed.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself. To do so, proceed as follows:
- Save all data
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of
- 18270 $MM_NUM_SUBDIR_PER_DIR
- 18310 $MM_NUM_DIR_IN_FILESYSTEM
- Power On
- Reload saved data

Program Continuation: Clear alarm with the RESET key. Restart part program

300413 Error when reading a file (%1, %2)

Parameters: %1 = Error code 1
%2 = Error code 2

Definitions: An attempt to read a data block, e.g. a drive boot file, from the file system has failed. The data block or the file system is damaged.

Reaction: Interface signals are set.
Alarm display.

Remedy: If the error occurred during power-up, i.e. it is probably connected to a drive boot file, delete all boot files and load them back into the control from the back-up copy.

Program Continuation: Clear alarm with the RESET key. Restart part program

300423 Measuring result could not be read (%1)

Parameters: %1 = Error code

Definitions: An attempt to read a measurement result has failed:
- Error code = 4: Not enough space for test result
- Error code = 16: Measurement not yet finished

Reaction: Interface signals are set.
Alarm display.

Remedy: Repeat measurement. Alter measuring time if necessary.

Program Continuation: Clear alarm with the RESET key. Restart part program

2.4 PROFIBUS alarms

380001 PROFIBUS/PROFINET: Startup error, reason %1 parameter %2 %3 %4.

Parameters: %1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Definitions: An error occurred during startup of the PROFIBUS/PROFINET master.
Overview: Cause of the error, Par 1, Par 2, Par 3:
- 01 = DPM version, DPM version, DPA version, --
- 02 = DPM ramp-up timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM ramp-up status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM ramp-up error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = DPM-PLL sync error, --, --, --
- 07 = Alarm queue too long, Actual number, Setpoint number, --
- 08 = Unknown client, Client ID, --, --
- 09 = Client version, Client ID, Client version, DPA version
- 10 = Too many clients, Client number, max. number of clients, --
- 11 = Log.basic address used several times; bus no.; slot no.; log.basic address --
- 20 = Slave/device address used several times, slave/device address --
- 21 = Slave/device address unknown, slave/device address, --
- 22 = Erroneous configuration telegram, slave/device address, error code, --
- 23 = OMI incompatible (data), drive version, CDA version, --, --
- 24 = OMI incompatible (driver), drive version, CDA version, --, --
- 25 = CPI initialization failed, error code, --, --, --
- 26 = DMA not active
- 27 = Reserved
- 28 = Reserved
- 29 = Reserved
- The 1000s digit of the error cause = number of the affected bus

Clients are the following components of the control system that use the PROFIBUS/PROFINET:
Client ID = 1: PLC
Client ID = 2: NCK
Possible causes are:
- Error in contents of SDB
- Corruption of parts of the system program
- Hardware defect on NC component

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Remedy for 1-11
1. Check the control project, check MD11240 $MN_PROFIBUS_SDB_NUMBER, and reload it when using a user-specific SDB.
2. If the error still occurs, back up data, and restart the control with the standard values as per the as-delivered condition.
3. In case of correct ramp-up, reload the user data stage by stage.
4. If the error still occurs during ramp-up with standard values, reboot the system from the PC card or update the software.
5. If the error still occurs, replace the hardware.
Remedy for 20-21
1. Check/correct the addresses of the connected slaves/devices.
Remedy for 22
See SINAMICS warning 1903 for a description of the meaning behind the error codes.
1. Control the SDB
- Check the type and length of the message frame
- Match slot assignment with P978
2. Evaluate the drive alarms/warnings
Remedy for 23-24
1. Software replacement required
Remedy for 25

1. Change the message frame type
2. Reduce the number of slots
3. Reduce the number of slaves/devices
4. Create a new SDB
5. Software must be replaced

If the error has still not been able to be rectified after this procedure, send the error text to the control manufacturer.

Program Continuation: Switch control OFF - ON.

380003 PROFIBUS/PROFINET: Operating error, reason %1 parameter %2 %3 %4.

Parameters:
%1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Definitions:
An operating error occurred on the PROFIBUS/PROFINET in cyclic mode.
Overview: Cause of the error, Par 1, Par 2, Par 3:
- 01 = unknown alarm, alarm class, logical address, --
- 02 = DPM cycle timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM cycle status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM cycle error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = Client not registered, client number, max. number of clients, --
- 06 = Synchronisation error, number of sync violation, --, --
- 07 = Spinlock timeout, PLC spinlock, NCK spinlock, --
- 1000s digit of the error cause = number of the affected bus

Alarm class: (see alarm 380 060)
The following can be primary causes:
- For error cause 01: Data transfer error on the PROFIBUS/PROFINET
- For error causes 02, 03, 04: Error in contents of SDB
- For error causes 02, 03, 04, 05, 07: Corruption of parts of system program
- For error cause 06: The PCI bus cycle does not match the expected rate, so synchronization is not possible. The correct PCI bus cycle must be entered.

The error can also be caused by a hardware problem on the MCI module.

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/PROFINET, assess the cable installation
- Check the terminating resistors of the PROFIBUS connectors (ON setting at ends of cables, otherwise OFF setting required)
- Check slave/device
- For error causes 02, 03, 04:
- Check SDB
- For error causes 02, 03, 04, 05, 07:
- Follow the procedure described for troubleshooting alarm 380 001
- For error cause 06:
- The correct PCI bus cycle must be entered.

If the error cannot be eliminated by this procedure, send the error text to the control system manufacturer.

Program Continuation: Clear alarm with the RESET key. Restart part program

380005 PROFIBUS/PROFINET: Bus %3 access conflict, type %1, counter %2

Parameters:
%1 = Conflict type
%2 = Serial number within a conflict sequence
%3 = Number of the affected bus

Definitions:
An access conflict occurred on the PROFIBUS/PROFINET in cyclic mode: An attempt has been made in the NCK to write data to the bus or to read from the bus while cyclic data transfer was active. This may lead to inconsistant data.
Type 1: NCK attempts to read data before the cyclic transfer has finished on the bus.

Type 2: The NCK has not finished writing its data when the cyclic transfer begins again. Counter %2 contains a serial number starting at 1. A maximum of 10 alarms are output in succession. If no conflicts occur in a DP cycle, the counter is reset and new alarms are output again on the next conflict.

Reaction: Alarm display.

Remedy: - Check the timing again, in particular ensure that the settings in MD10050 $MN_SYSCLOCK_CYCLE_TIME and MD10062 $MN_POSCTRL_CYCLE_DELAY are correct:
MD10062 $MN_POSCTRL_CYCLE_DELAY must be larger for type 1.
MD10062 $MN_POSCTRL_CYCLE_DELAY must be smaller for type 2.
- If alarm-free operation cannot be achieved with any MD10062 $MN_POSCTRL_CYCLE_DELAY setting, MD10050 $MN_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/PROFINET: Bus %3 SDB %4 error %1 source %2

Parameters: %1 = Error
%2 = SDB source
%3 = Bus number
%4 = SDB number

Definitions: Error in SDB for configuring PROFIBUS/PROFINET.
Causes of the error:
- 01 = SDB does not exist in source.
- 02 = SDB from source is too large.
- 03 = SDB from source cannot be activated.
- 04 = Source is empty.
- 05 = Source is not present.

SDB source:
- 99 = Passive file system: _N_SDB_DIR
- 100 = CF card: /siemens/sinumerik/sdb/...
- 101 = CF card: /addon/sinumerik/sdb/...
- 102 = CF card: /oem/sinumerik/sdb/...
- 103 = CF card: /user/sinumerik/sdb/...

Reaction: PROFIBUS/PROFINET is inactive or working with the default SDB.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check the setting of MD 11240 $MN_PROFIBUS_SDB_NUMBER.
- If source = 100: Check directory _N_SDB_DIR in the passive file system.
- If source = 103-106: Check directories on CF card

Program Continuation: Switch control OFF - ON.

380022 PROFIBUS/PROFINET: Configuration of DP master bus %1 has been changed

Parameters: %1 = Number of the affected bus

Definitions: The PROFIBUS configuration on the DP master was changed during operation, e.g. by downloading a new hardware configuration via STEP 7. As the cycle data may also have changed, operation cannot be continued, and a warm start is required.
If the master functionality is within the PLC (as on the 840Di sl), the PLC will have been stopped for the download, and alarm 2000 (PLC sign-of-life) output.

Reaction: Channel not ready.
NC Start disable in this channel.
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/PROFINET: Bus %3, configuration error %1, parameter %2

Parameters: %1 = Cause of the error
%2 = Parameter
%3 = Number of the affected bus

Definitions: The PROFIBUS/PROFINET was not generated in the SDB in accordance with the configuration specifications of the NC in use.
Overview: Cause of the error, Par 1:
- 01 = SDB contains slave/device without diagnostics slot, slave/device address
- 02 = SDB contains too many slot entries, identifier
- 03 = SDB contains no equidistance data, no function.
- 04 = PNIO: SDB contains different Tdp (also TDC) on a device
- 05 = PNIO: SDB contains different Tmapc (also CACF) on a device
- 06 = PNIO: SDB contains different TI on a device
- 07 = PNIO: SDB contains different TO on a device
- 08 = PNIO: SDB contains device numbers that are too high (with values higher than 126)
- 09 = SDB contents are transferred segmented (too many slots/frames)
- 20 = SDB contains too many slaves/devices, quantity.
- 21 = SDB missing or contains invalid data, error code.
- 22 = SDB configuration data incorrect, slave/device address, error code
- 23 = Reserved
- 24 = Reserved
- 25 = Reserved
- 26 = Reserved
- 27 = Reserved
- 28 = Reserved
- 29 = Reserved

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check that the corresponding SDB
- Contains a diagnostic slot for every slave/device, and
- Contains only slave/device entries relevant to the application.
In general, it is possible to include a superset of slaves/device in the SDB that are partially relevant for different end versions of the product. However, this overloads the NC memory and runtime capacity, and should therefore be avoided in general.
If this alarm occurs, it is necessary to reduce the SDB to a minimum.
If the code for the error cause is 03, check that equidistance is activated in the SDB (e.g. using STEP 7 HW config).
If the alarm continues to occur, please send the error text to the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380050 PROFIBUS/PROFINET: 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:
- MD13050 $MN_DRIVE_LOGIC_ADDRESS[0] - MD13050 $MN_DRIVE_LOGIC_ADDRESS[n-1] : n = highest axis index on control system
- MD12970 $MN_PLC_DIG_IN_LOGIC_ADDRESS, MD12971 $MN_PLC_DIG_IN_NUM : PLC address area for digital inputs
- MD12978 $MN_PLC_ANA_IN_LOGIC_ADDRESS, MD12979 $MN_PLC_ANA_IN_NUM : PLC address area for analog inputs

If no inconsistencies can be found in the parameters, compare these machine data with the configuration in SDB (STEP 7 project). 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/PROFINET: 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:
- MD13050 $MN_DRIVE_LOGIC_ADDRESS[0] - MD13050 $MN_DRIVE_LOGIC_ADDRESS [n-1] : n = highest axis index on control system
- MD12974 $MN_PLC_DIG_OUT_LOGIC_ADDRESS, MD12975 $MN_PLC_DIG_OUT_NUM : PLC address area for digital outputs
- MD12982 $MN_PLC_ANA_OUT_LOGIC_ADDRESS, MD12983 $MN_PLC_ANA_OUT_NUM : PLC address area for analog outputs

If no inconsistencies can be found in the parameters, compare these machine data with the configuration in the SDB (STEP 7 project). 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/PROFINET: Alarm %1 on logical address %2 from unassigned slave/device

Parameters: %1 = Alarm class
%2 = Logical address

Definitions: SDB contains a slave/device which is not assigned in the NC via the MD parameters (see also alarm 380050/051). The slave/device is however connected to the PROFIBUS/PROFINET, and has reported an alarm.
Alarm class:
- 01 = Station return (or arrival)
- 02 = Station failure

Display alarm, further operation with the NC is possible.

Reaction: Alarm display.

Remedy:
- Enter machine data or
- Modify SDB or
- Disconnect the slave/device from the PROFIBUS/PROFINET or
- Acknowledge the alarm.

Program Continuation: Clear alarm with the Delete key or NC START.

380070 PROFIBUS/PROFINET: No input slot available for basic address %1 (length %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
Length=1 indicates a digital input.
Length=2 indicates a analog input.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTIN.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTIN.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380071 PROFIBUS/PROFINET: No output slot available for basic address %1 (size %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
For length =1 it is a digital output,
For length =2 it is an analog output.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380072 PROFIBUS/PROFINET: Output slot for basic address %1 (size %2) not allowed

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Definitions: An incorrect logical base address was set for a digital or analog output, the area is resides in the access range of the PLC (PIQ, base addresses < 256).
For length =1 it is a digital output,
For length =2 it is an analog output.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Only use addresses outside the PLC process image (e.g. >= 256) for output slots.
Enter correct basic addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Program Continuation: Switch control OFF - ON.

380075 PROFIBUS/PROFINET: DP I/O failure bus %2 slave/device %1

Parameters: %1 = Slave/device address
%2 = Number of the affected bus

Definitions: Failure of a PROFIBUS/PROFINET slot used by the NCK for digital or analog I/Os.

Reaction: Alarm display.

Remedy: Check that the slave/device is operating correctly (all slaves/devices must be included in the bus, green LEDs).

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380076 PROFIBUS/PROFINET: No DO1 message frame: Bus %2 slave/device %1

Parameters: %1 = Slave/device address
%2 = Number of the affected bus

Definitions: Note for the system startup engineer: A PROFIBUS slave/PROFINET device used as an NCK drive does not have a valid DO1 message frame assignment (see MD13120 $MN_CONTROL_UNIT_LOGIC_ADDRESS with the STEP 7 configuration).
This alarm is indicating, among other things, that the alarm time-of-day synchronization is not working between the controller and this slave/device.

Reaction: Alarm display.

Remedy: Enter a valid value in MD13120 $MN_CONTROL_UNIT_LOGIC_ADDRESS.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380500 PROFIBUS/PROFINET: Fault on drive %1, code %2, value %3, time %4

Parameters: %1 = Axis
%2 = Fault code of drive (P947(/945)/P824)
%3 = Fault value of drive ((P949/P826)
%4 = Fault time of drive (P948/P825)

Definitions: Contents of fault memory of assigned drive.

Reaction: Alarm display.

Remedy: See drive documentation for fault codes/fault values.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380501 PROFIBUS/PROFINET: Fault on bus, slave/device, DO ID %1, code %2, value %3, time %4

Parameters: %1 = 8 bit bus number, 8 bit slave/device number, 16 bit DO ID
%2 = Fault code of drive (P947)
%3 = Fault value of the drive (P949)
%4 = Fault time of the drive (P948)

Definitions: Contents of the fault memory of the assigned slave/device.

Reaction: Alarm display.

Remedy: See drive documentation for fault codes/fault values.

Program Continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380502 PROFIBUS/PROFINET: Bus %1, slave/device %2 configuration changed

Parameters: %1 = Bus number
%2 = Slave/device address

Definitions: The bus configuration has changed.
Causes:
- Initial start-up
- New slave/device recognized on the bus

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.

380503 PROFIBUS/PROFINET: Bus %1 configuration changed

Parameters: %1 = Bus number

Definitions: A new SDB with a modified configuration has been provided.
The new settings will be activated only at the next bus power up.

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.

2.5 PLC alarms

400000 PLC STOP %1

Definitions: PLC not in cyclic mode. Travel with the machine is not possible.
%1: 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.

400001 System error %2

Definitions: %1 :Type number
With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reaction: PLC Stop

Remedy: Notify Siemens of this error together with the error message.

Program Continuation: Switch control OFF - ON.

400002 System error %1

Definitions: %1 :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.

Program Continuation: Switch control OFF - ON.

400004 Code error: %2 network %1

Definitions: %1 :Network number
%2 :Internal error code, module type
The user program contains an operation which is not supported by the control.

Reaction: PLC Stop

Remedy: Modify and reload user program.

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: %2 network %1

Definitions: %1 :Network number
%2 :Module type

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.

400008 Programming tool - version is not compatible %1 %2

Definitions: %1 :Programming tool version
This version is not compatible with the product version 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: %2 network %1

Definitions: %1 :Network number
%2 :Module type
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 %2 network %1

Definitions: Check user program in the specified network.
%1Network number, module ID
%2 = 1:Division by zero using fixed-point arithmetic
2:Floating-point arithmetic

Reaction: PLC Stop

Remedy: Change user program.

Program Continuation: Switch control OFF - ON.

400011 Maximum number of subroutine levels exceeded: %2 network %1

Definitions: %1Network number
%2Module ID
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

Definitions: %1: 1PROFIBUS DP power up interrupted
2Software versions of NC and PLC do not match
3Number of slots per function exceeded
4PROFIBUS DP server not ready

Reaction: PLC Stop

Remedy: Types 1 to 3:Report error to Siemens
Type 4:802D sl – Check and/or replace PCU hardware and/or check MD 11240

Program Continuation: Switch control OFF - ON.

400015 PROFIBUS DP: I/O defect: log adr. %1 bus/station: %2

Definitions: The PLC-AWP is using I/O addresses which are not available.
%1Logical I/O address

	%2Bus number/station number Causes of error: Bus peripheral has no voltage Bus address set incorrectly Bus connection faulty Active MD 11240 (SDB configuration) is set incorrectly
Reaction:	PLC Stop
Remedy:	Rectify the error using the error cause
Program Continuation:	Switch control OFF - ON.

2.6 SINAMICS alarms

A detailed description of the SINAMICS alarms is provided in the following publication:

LH1, SINAMICS_S Parameter Manual, Chapter: Faults and Warning

Structure of the SINAMICS alarms

Each alarm (fault or warning) consists of a number, local ID (optional) and alarm text.

Number range

Numbers range from 200 000 to 299 999.

Notation

In the SINAMICS_S List Manual, faults and warnings start with letter "F" and a following five-digit number.

Example:

You can find a description of alarm number **207016** in the SINAMICS_S List Manual under No. **F07016**.

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	-

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.

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 -

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

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 -

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

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: -

No. 33, STARTSIG

Explanation Start selected processing (VDI signal: NC Start).

Cause
1. Process change active (operating mode change, activate/deactivate digitizing/overstore)
2. An alarm reaction is pending which prevents a start or forces braking.
3. A process is running (NC program, block search, loading machine data)

Remedy Execute condition for clearing alarm

No. 34, 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 HMI)
(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 -

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 -

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 -

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 -

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

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 -

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: -

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 -

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 -

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 -

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 HMI; 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	-

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 -

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	-

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).

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	-

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

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	-

No. 133

Explanation	Refusal of PI service _N_NCKMOD (BIT 1)
Cause	-
Remedy	-

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

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 HMI.

Alarm display

Name STOPBYALARM

Effect Ramp stop of all channel axes.

– NC stop for alarm

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 HMI. 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.

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.

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.

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

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

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

T	Tool
TO	Tool offset
ZOA	Zero Offset Active: Memory area

Index

Number

To Siemens AG I DT MC MS1 P.O. Box 3180 91050 ERLANGEN, GERMANY Fax: +49 (0) 9131 98 2176 [documentation]	**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.: 6FC5398-2CP10-2BA0 06/2009 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