Diagnostics Guide 10/2002 Edition

sinumerik

SIEMENS

SINUMERIK 802D

SIEMENS

SINUMERIK 802D

Diagnostics Guide

User Documentation

Valid for

Control	*Software version*
SINUMERIK 802D	as of 1

10.02 Edition

SINUMERIK® Documentation

Printing history

The editions listed below have been published prior to the current edition.

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

Edition	Order No.	Note
11.00	6FC5698-2AA20-0BP0	A
10.02	6FC5698-2AA20-0BP1	C

Tested Siemens quality for software and training
to DIN ISO 9001, Reg. No. 2160-01

This publication was created using WinWord V 8
and Designer V 6.0.

Other functions not described in this documentation can possibly also be performed on the control system. However, the customer is not entitled to demand these functions when the new equipment is supplied or servicing is carried out.

Although we have checked the contents of this publication for agreement with the hardware and software described, since differences cannot be totally ruled out.The information in this publication is checked at regular intervals and necessary corrections will be included in the next releases.Suggestions for improvement are always welcome.Subject to change without prior notice.

Order No. 6FC5698-2AA20-0BP1
Printed in the Federal Republic of Germany

Siemens-Aktiengesellschaft

Preface

The present description is intended to be used as a reference manual. It allows the operator on the machine tool to:

- access special cases in the operation of the machine correctly;
- to learn the response of the system/installation to the particular special case;
- to use the possibilities of continuing work after the special case;
- to follow notes for further references.

Scope

Object of the present description are the alarms occurring in the areas NC kernel (NCK), Profibus, cycles and PLC.
Further alarms may occur in the HMI area (Human Machine Interface). They are announced to the user by self-explaining alarm lines displayed on the operator panel. These alarms are not part of this Diagnostics Guide.
For special cases in conjunction with the integrated PLC, please refer to the relevant SIMATIC S7-200 system documentation.

Sorting

The alarms are sorted in the Diagnostics Guide by ascending alarm numbers with gaps between them.

Safety

Danger

Please always check the situation of the particular system/installation carefully on the basis of the alarms occurred. Eliminate the causes for the occurrence of the alarms and acknowledge them as described. Otherwise, danger will result for machine, workpiece, saved settings and - under certain circumstances - even for your health.

NC alarms

Table 1_1 Alarm number ranges

000 000 - 009 999	General alarms	
010 000 - 019 999	Channel alarms	
020 000 - 029 999	Axis / spindle alarms	
030 000 - 099 999	Functional alarms	
060 000 - 064 999	SIEMENS cycle alarms	
065 000 - 069 999	User cycle alarms	

HMI alarms/messages

Table 1_2 Alarm number ranges (cont'd.)

100 000 - 100 999	Basic system	HMI0
101 000 - 101 999	Diagnosis	
102 000 - 102 999	Services	
103 000 - 103 999	Machine	
104 000 - 104 999	Parameters	
105 000 - 105 999	Programming	
106 000 - 106 999	Reserve	
107 000 - 107 999	OEM	
110 000 - 110 999		Reserved
120 000 - 120 999		Reserved

PLC alarms/messages

Table 1_4 Alarm number ranges (cont'd.)

400 000 - 499 999	General alarms	
700 000 - 799 999	User area	

Contents

Alarms 1

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 errors 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:**

Hotline Germany

Siemens AG, A&D MC
Tel. +49 (0) 180 525 80 08
Fax +49 (0) 180 525 80 09

Hotline China

Siemens Numerical Control Ltd.
Development & Engineering Division
Tel. (025) 2 18 18 88 (Ext. 305)
Fax (025) 2 18 16 66

1.1 Overview of NC alarms

2000

Sign-of-life monitoring PLC

Explanation	The PLC must provide a sign of life within a defined period of time. If not, this alarm is generated.
Reaction	NC Start inhibited. NC not ready. NC Stop at alarm. Alarm display. Interface signals are set.
Remedy	This alarm also occurs as a consequence of PLC Stop. (PLC Stop via Programming Tool, PLC Stop from start-up switch, PLC Stop caused by an alarm) If none of the above mentioned cases exists, please contact the hotline indicated in the beginning of this publication and specify the operating system error number.
Program continuation by	Power ON

2001

PLC not booted

Explanation	The PLC must provide at least one sign of life within the defined period of time after POWER ON.
Reaction	NC Start inhibited. NC not ready. NC Stop at alarm. Alarm display. Interface signals are set.
Remedy	Contact the hotline indicated in the beginning of this publication.
Program continuation by	Power ON

2140

The current position of the service switch will clear the SRAM with the next POWER ON (general reset active)

Explanation	The initialization switch is currently set to "General reset". As a consequence, the SRAM of the module is cleared with the next module reset; as a result, the NC data memory is lost.
Reaction	Alarm display. Interface signals are set. NC not ready.
Remedy	Reset the initialization switch back to "1".
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

3000

EMERGENCY STOP

Explanation	The EMERGENCY STOP request is present at the NC/PLC interface (V 26000000.1).
Reaction	NC Start inhibited. NC not ready

NC Stop at alarm
Alarm display
Interface signals are set.

Remedy

Check whether an EMERGENCY STOP cam has been approached or an EMERGENCY STOP button has been tripped. Check the PLC user program.

Correct the EMERGENCY STOP cause and acknowledge EMERGENCY STOP via the PLC/NC user interface (V 26000000.2).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

4000

Channel %1 machine data %2 contains a gap in the axis assignment

Explanation

%1 = channel number
%2 = string: MD identifier

The assignment of a machine axis to a channel using the MD 20070 **AXCONF_MACHAX_USED** must be carried out without gaps. Any gaps will be detected when the system boots (POWER ON) and be displayed as an alarm.

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Please inform the authorized personnel/customer service.

Configure MD 20070 **AXCONF_MACHAX_USED** for the axis assignment of the channel without gaps, i.e. with ascending axis indices, a machine axis has to be assigned until zero (no machine axis) is entered for the first time. In this case, all MDs that have higher indices must also be loaded with zero.

The sequence order of the machine numbers is not relevant.

Channel Index	1st Channel	Machine Axis Number
0	1	AXCONF_MACHAX_USED [CH1, AX1] = 1
1	2	AXCONF_MACHAX_USED [CH1, AX2] = 2
2	3	AXCONF_MACHAX_USED [CH1, AX3] = 3
3	4	AXCONF_MACHAX_USED [CH1, AX4] = 4
4	5	AXCONF_MACHAX_USED [CH1, AX5] = 5

Assignment of the channel axes to the machine axes

Program continuation by

Power ON

4002

Channel %1 machine data %2[%3] contains an axis not defined in the channel

Explanation

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

Only axes activated in the channel via MD 20070 **AXCONF_MACHAX_USED** [kx]=m
may be declared geometry axes via
MD 20050 **AXCONF_GEOAX_ASSIGN_TAB [gx]=k**.

gx ...	Geometry axis index	k ...	Channel axis no.
kx ...	Channel axis index	m ...	Machine axis no.

MD 20050 **AXCONF_GEOAX_ASSIGN_TAB** (contains channel axis no. k)		MD 20070 **AXCONF_MACHAX_USED** (contains machine axis no. m)	
Geometry Axis Index	1. Channel	Channel Axis Index	1. Channel
0	1	0	1
1	2	1	2
2	3	2	3
		3	4
		4	5

Assignment of the geometry axes to the channel axes

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Please inform the authorized personnel/customer service.
Check MD 20050 **AXCONF_GEOAX_ASSIGN TAB** and
MD 20070 **AXCONF_MACHAX_USED** and correct them if necessary.

The basis is the MD array MD 20070 **AXCONF_MACHAX_USED** in which the machine axis number to be controlled by this channel is entered channel-specifically. The resulting "channel axes" are assigned a name by MD 20080 **AXCONF_CHANAX_NAME_TAB** and assigned to a geometry axis via MD 20050 **AXCONF_GEOAX_ASSIGN_TAB** by entering the relevant channel axis in this MD array for each geometry axis.

Program continuation by

Power ON

4004

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

Explanation

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

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

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Correct MD 20050 **AXCONF_GEOAX_ASSIGN_TAB**.

Program continuation by

Power ON

Power ON

4010

Machine data %1[%2] contains an illegal identifier

Explanation

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

When defining the names for: machine axes, one of the following syntax rules has been violated for the identifier to be entered:

1. The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension.
2. The identifier must start with 2 random uppercase letters, but not with the $ character (reserved).
3. The identifier must not be a vocabulary word of the NC language (e.g. SPOS).

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Please inform the authorized personnel/customer service.
Enter the identifier for user-defined names in the displayed MD using the correct syntax.

Machine axes: MD 10000 **AXCONF_MACHAX_NAME_TAB**

Program continuation by

Power ON

4011

Channel %1 Machine data %2[%3] contains an illegal identifier

Explanation

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

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

1. The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension.
2. The identifier must start with 2 random uppercase letters, but not with the $ character (reserved for system variables).
3. The identifier must not be a vocabulary word of the NC language (e.g. SPOS).

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Please inform the authorized personnel/customer service.
Enter the identifier for user-defined names in the displayed MD using the correct syntax.

Geometry axes: MD 20060 **AXCONF_GEOAX_NAME_TAB**
Channel axes: MD 10000 **AXCONF_MACHAX_NAME_TAB**

Program continuation by

Power ON

4020

Identifier %1 used repeatedly in machine data %2

Explanation

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

When defining the names in the NC tables (arrays) for **machine axes**, an identifier was used which already exists in the control system.

Reaction

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

NC Start inhibited.
NC Stop at alarm.

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

Program continuation by: The RESET key must be used in all channels of this mode group to cancel this alarm.

4021

Channel %1 Identifier %2 used repeatedly in machine data %3

Explanation: %1 = channel number
%2 = string: Identifier
%3 = string: MD identifier

When defining the names in the channel-specific tables for **geometry axes and channel axes**, an identifier was used which already exists in the control system.

Reaction: Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

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

Program continuation by: Power ON

4030

Channel %1 Missing identifier in machine data %2[%3]

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

Due to the axis configuration in MD 20070 **AXCONF_MACHAX_USED** and MD 20050 **AXCONF_GEOAX_ASSIGN_TAB,** an axis identifier is expected for the MD displayed.

Reaction: Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy: Please inform the authorized personnel/customer service.
Check the axis configuration and enter the missing axis identifier in the MD or - if you wish the axis to be omitted - specify machine axis 0 for this channel axis in MD 20070 **AXCONF_MACHAX_USED**. If the axis is a geometry axis which will not be used (with exclusively 2-axis machining, e.g. with turning machines), the channel axis 0 must additionally entered in MD 20050 **AXCONF_GEOAX_ASSIGN_TAB** for the appropriate geometry axis.

Program continuation by: Power ON

4032 Channel %1 Wrong identifier for transverse axis in %2

Explanation

%1 = channel number
%2 = string: MD identifier

Due to the axis configuration in MD 20150 **GCODE_RESET_VALUES** or MD 20100 **DIAMETER_AX_DEF**, a transverse axis identifier is expected in the specified place.

Reaction

Alarm display
Interface signals are set
NC Stop at alarm
NC not ready
NC Start inhibited

Remedy

Please inform the authorized personnel/customer service.
Add the correct identifier.

Program continuation by

Power ON

4040 Channel %1 axis identifier %2 inconsistent with machine data %3

Explanation

%1 = channel number
%2 = string: Axis identifier
%3 = string: MD identifier

The use of the axis identifier specified in the displayed MD is not consistent with the axis configuration of the channel specified in MD 20070 **AXCONF_MACHAX_USED** and
MD 20050 **AXCONF_GEOAX_ASSIGN_TAB**.

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Please inform the authorized personnel/customer service.
Check the identifiers used in MD 10000 **AXCONF_ MACHAX_NAME_TAB,** MD 20080 **AXCONF_CHANAX_NAME_TAB** and/or MD 20050 **AXCONF_GEOAX_NAME_TAB** and correct them if necessary.

Program continuation by

Power ON

4050 NC code identifier %1 was not reconfigured to %2

Explanation

%1 = string: Old identifier
%2 = string: New identifier

The NC code could not be renamed for one of the following reasons:

- The old identifier does not exist any more.
- The new identifier is assigned to a different type area.

NC codes/vocabulary words can be reconfigured via machine data provided that the type area is not left.

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

Type 2: Known G codes: CIP, TRANS, ...

Reaction

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy
Please inform the authorized personnel/customer service.
Correct MD 10712 **NC_USER_CODE_CONF_NAME_TAB** (protection level 1).

The list must be structured as follows:

Even address: Identifier to be changed
followed by an odd address: New identifier

e.g.: NC_USER_CODE_CONF_NAME_TAB [10] = "ROT"
NC_USER_CODE_CONF_NAME_TAB [11] = " "
will cancel the ROT function in the control system

Program continuation by
Power ON

4060

Default machine data have been loaded

Explanation
Booting with the default values by:

- operator action (e.g. start-up switch)
- MD 11200 **INIT_MD**
- loss of the retentive data
- operator action - booting with saved data, without saving the data first

Reaction
Alarm display.

Remedy
After the default machine data have been loaded, the particular MD for your particular system/installation must be entered/loaded.

Program continuation by
Use the "Cancel" key to cancel the alarm. Reload your own machine data.

4062

Data backup copy has been loaded

Explanation
The used data saved to flash have been loaded into the SRAM.

Reaction
Alarm display.

Remedy
Reload your own machine data.

Program continuation by
Use the RESET key to cancel the alarm.

4065

Battery-backed memory is restored from backup copy (risk of data loss !)

Explanation
When booting, a possible inconsistency has been detected in the battery-backed memory.

The battery-backed memory was initialized using the last backup copy. As a result, the changes in the battery-backed memory carried out since the last update of the backup copy are lost. The cause of this procedure is to be searched for in exceeding of the buffer time. Please make sure that the required on-time of the control system, which is specified in your Start-up Guide, is not exceeded.

The current backup copy of the battery-backed memory was created using the internal data backup carried out last using the softkey "Save data" in the HMI.

Reaction
Alarm display
Interface signals are set.
NC Start inhibited

Remedy
Restart the control system.

Program continuation by

4070 Unit machine data changed

Explanation

The control system uses internal physical units (mm, degrees, s, for distances to go, velocities, accelerations and the like). With programming or data back-up, some of these values are input/output using other units (rpm, m/s2, etc.).

The conversion is carried out using scaling factors which can be entered (system-specific MD array 10230 **SCALING_FACTORS USER_DEF**[n] (n ... index numbers 0 ... 10) if the relevant masking bit is set to "1".

If the masking bit is set to "0", the scaling is carried out using the internal default factors.

The following machine data influence the scaling of other MDs:

MD 10220 **SCALING_USER_DEF_MASK**

MD 10230 **SCALING_FACTORS_USER_DEF**

MD 10240 **SCALING_SYSTEM_IS_METRIC**

MD 10250 **SCALING_VALUE_INCH**

MD 30300 **IS_ROT_AX**

After these data have been changed, the NC must be rebooted. Only then the input of dependent data will be executed correctly.

Reaction

Alarm display.

Remedy

Please inform the authorized personnel/customer service.
If the alarm has been displayed after downloading a consistent MD file, the download operation must be repeated and the NC is rebooted. (Scaling-dependent machine data in the file are always put in front of the scaling factors).

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

4075 Machine data %1 (and possible further) not changed due to missing access rights %2

Explanation

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

When executing a TOA file, it has been tried to program a data whose protection level is higher than the access authorization currently set at the control system.
The relevant data has not been programmed.
This alarm is only generated with the first write access violation detected.

Reaction

Alarm display.

Remedy

Either set the required access level via the password or delete the appropriate machine data from the MD file.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

4076 %1 Machine data could not be changed with access right %2

Explanation

%1 = number of MD
%2 = set access right

When executing a TOA file, it has been tried to program a data whose protection level is higher than the access authorization currently set at the control system.
The appropriate data have not been programmed.
This alarm is issued when acknowledging alarm 4075. It can only be canceled with POWER ON.

Reaction	Alarm display.
Remedy	Either set the required access level via the password or using the keyswitch or delete the appropriate machine data from the MD file.
Program continuation by	Power ON

4077

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

Explanation

%1 = new value of the machine data
%2 = machine data number
%3 = number of bytes requested in excess
%4 = type of memory

It was tried to assign the specified memory-configuring machine data a new value.

The change will not be executed, since it would result in deleting the user memory. The change requires more user memory than provided.

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

The fourth parameter specifies the type of the memory whose limit is exceeded:

"D" stands for the dynamic or non-backed user memory (there are, e.g. the LUD variables, this also includes the IPO buffer size). The size of this memory type is defined by the current memory configuration and by the value of MD18210 **MM_USER_MEM_DYNAMIC**.

"S" stands for the static or battery-backed user memory (typically, there are the part programs, but also the compensation data, R parameters, tool data); this memory type is also defined by the current memory configuration and by the value of MD 18230 **MM_USER_MEM_BUFFERED**.

Reaction	Alarm display.
Remedy	If the change was not desired, then simply continue. In this case, the alarm will not have any negative influence. How this error is corrected depends on the access right and on the current NC memory configuration: The change as planned to be done is not possible in this way -> try once more using a smaller value. When doing so, watch how the value of the byte number changes. Buying more memory? This possibility depends on the model you are using. The NC user memory is possible set smaller than possible. Provided the appropriate access right, the MD (see above) can be changed.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

4090

Too many errors when booting

Explanation	More than <n> errors have occurred when the control system has booted.
Reaction	Alarm display. NC Start inhibited
Remedy	Set the machine data correctly.
Program continuation by	

4110 IPO cycle factor increased to %1 ms

Explanation: %1 = string (new IPO cycle)

The IPO cycle divider was set to a value which was no integer multiple of the position control divider.
The divider (MD 10070 IPO_SYSCLOCK_TIME_RATIO) was increased.
In the case of systems working with Profibus DP, IPO_SYSCLOCK_TIME_RATIO has been modified due to the changed DP cycle (MD 10050 SYSCLOCK_CYCLE_TIME) in SDB1000.

Reaction: Alarm display.

Remedy: The machine data 10070 IPO_SYSCLOCK_TIME_RATIO was adapted accordingly.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

4111 Increase PLC clock to %1 ms

Explanation: The PLC clock divider was set to a value which was no integer multiple of the IPO clock divider.
The divider (MD **10 074 PLC_IPO_TIME_RATIO**) was increased.

Reaction: Alarm display.

Remedy: Adapt the machine data accordingly.

Program continuation by: Power ON

4112 Servo clock changed to %1 ms

Explanation: %1 = string (new servo cycle)

In the case of systems working with Profibus DP, IPO10060 _SYSCLOCK_TIME_RATIO has been modified in SDB1000 due to the changed DP cycle (MD 10050 SYSCLOCK_CYCLE_TIME).

Reaction: Alarm display.

Remedy: The machine data 10060 POSCTRL_SYSCLOCK_TIME_RATIO was adapted accordingly.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

4113 Sysclock changed to %1 ms

Explanation: %1 = string (new PLC cycle)

The MD 10050 SYSCLOCK_CYCLE_TIME has been modified due to the changed DP clock in SDB1000.

Reaction: Alarm display.

Remedy: The machine data 10050 IPO_SYSCLOCK_CYCLE_TIME was adapted accordingly.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

4114 Error in DP clock of SDB1000

Explanation: %1 = string (new PLC cycle)

The DP clock in SDB1000 is faulty and can no longer be set.
The default value of $MN_SYSCLOCK_CYCLE_TIME will be set.

Reaction: Alarm display.

Remedy	Correct SDB1000 accordingly.
Program continuation by	Power ON

4150 Channel %1 Invalid M function subroutine call configured

Explanation %1=channel number	In MD 10715 **M_NO_FCT_CYCLE** for configuring the subroutine call via M function, an M function was specified which is occupied by the system and cannot be replaced by a subroutine call (M0 to M5, M17, M19, M30, M40 to M45, M70). With the external language active, M96 – M99 are also disabled
Reaction	Alarm display. NC Start inhibited Interface signals are set NC not ready NC Stop at alarm
Remedy	Configure an M function in MD 10715 **M_NO_FCT_CYCLE**, which is not occupied by the system (M0 to M5, M17, M19, M30, M40 to M45, M70).
Program continuation by	

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

Explanation	The function "Block display with absolute values" was configured illegally: - An illegal block length was set via $MC_MM_ABSBLOCK: When booting, the machine data is checked for the following range of values: 0, 1, 128 ... 512 - An illegal display range was set via $MC_MM_ABSBLOCK_BUFFER_CONF[]. When booting, the machine data is checked for the following upper/lower limits: 0 <= $MC_MM_ABSBLOCK_BUFFER_CONF[0] <= 8 0 <= $MC_MM_ABSBLOCK_BUFFER_CONF[1] <= ($MC_MM_IPO_BUFFER_SIZE + $MC_MM_NUM_BLOCKS_IN_PREP). In case of violation of the limits, alarm 4152 is issued.
Reaction	Alarm display. Interface signals are set. Channel not ready. NC Stop at alarm. NC Start inhibited.
Remedy	Dimension the block length/display range within the permitted limits.
Program continuation by	Power ON

4160 Channel %1 Invalid M function number configured for spindle switching

Explanation	%1=channel number In MD 20094 **SPIND_RIGID_TAPPING_M_NR** for configuring the M function number for switching the spindle to the axis mode, an M function was specified which is occupied by the system and cannot be used for switching (M1 ... M5, M17, M30, M40 ... M45).
Reaction	Alarm display. Interface signals are set. NC not ready. NC Start inhibited NC Stop at alarm
Remedy	Configure an M function in MD 20094 **SPIND_RIGID_TAPPING_M_NR**, which is not occupied by the system (M1 to M5, M17, M19, M30, M40 to M45, M70).

Program continuation by

4182

Channel %1 Illegal M auxiliary function number in %2%3; MD reset

Explanation

%1 = channel number
%2 = machine axis identifier
%3 = MD index if necessary

A number which is occupied by the system and cannot be used for an assignment was specified for configuring an M function.
(M0 ... M5, M17, M30, M40 ... M45 and - when working with ISO dialect - also M98,M99).
The value used by the user has been reset by the system to the default value.

Reaction

Alarm display.
Interface signals are set.
Channel not ready.
NC Start inhibited in this channel
NC Stop at alarm.

Remedy

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

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

4183

Channel %1 M auxiliary function number %2 used repeatedly (%3 and %4)

Explanation

%1 = channel number
%2 = M auxiliary function number
%3 = machine axis data identifier
%4 = machine data identifier

A number in the specified machine data was used repeatedly for configuring an M function.

Reaction

Alarm display.
Interface signals are set.
Mode group not ready.
Channel not ready.
NC Start inhibited in this channel
NC Stop at alarm.

Remedy

Check the specified machine data and provide for an unambiguous assignment of the M auxiliary function numbers.

Program continuation by

Power ON

4200

Channel %1 Geometry axis %2 may not declared a rotary axis

Explanation

%1 = channel number
%2 = axis name:

The geometry axes constitute a Cartesian coordinate system; therefore, the declaration of a geometry axis as a rotary axis will result in a definition conflict.

Reaction

NC not ready.
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Please inform the authorized personnel/customer service.
Remove the rotary axis declaration of this machine axis.

To do so, determine the geometry axis index for the displayed geometry axis via the machine data array 20060 **AXCONF_GEOAX_NAME_TAB**. The channel axis number is stored in the MD array 20050 **AXCONF_GEOAX_ASSIGN_TAB** via the same index. The channel axis number minus 1 results in the channel axis index under which the machine axis number is found in the MD array 20070 **AXCONF_MACHAX_USED**.

Program continuation by: Power ON

4210 Channel %1 spindle %2 Rotary axis declaration missing

Explanation: %1 = channel number
%2 = axis name, spindle number

If you wish to operate a machine axis as a spindle, this machine axis must be declared as a rotary axis.

Reaction: NC not ready.
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

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

Program continuation by: Power ON

4215 Channel %1 spindle %2 Modulo axis declaration missing

Explanation: %1 = channel number
%2 = axis name, spindle number

The spindle functionality requires a modulo axis (positions in [deg]).

Reaction: NC not ready.
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy: Please inform the authorized personnel/customer service.
Set MD 30310 **ROT_IS_MODULO**.

Program continuation by: Power ON

4220 Channel %1 spindle %2 declared repeatedly

Explanation: %1 = channel number
%2 = axis name, spindle number

The spindle exists in the channel repeatedly.

Reaction: Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy: Please inform the authorized personnel/customer service.
The spindle axis number is stored in the axis-specific MD array 35000 **SPIND_ASSIGN_TO_MACHAX**. Which channel this machine axis / spindle will be assigned can be seen from the machine axis index. (The machine axis number is to be found in the MD array 20070 **AXCONF_MACHAX_USED).**

Program continuation by	Power ON

4225 Channel %1 axis %2 Rotary axis declaration missing

Explanation	%1 = channel number %2 = axis name, axis number The modulo functionality requires a rotary axis (positions in [deg]).
Reaction	NC not ready. NC Stop at alarm. NC Start inhibited. Alarm display. Interface signals are set.
Remedy	Please inform the authorized personnel/customer service. Set MD 30300 **IS_ROT_AX**.
Program continuation by	Power ON

4230 Channel %1 In current channel status, data cannot be changed from external source

Explanation	%1=channel number This data may not be entered during part program execution (e.g. setting data for the spindle speed limitation or for the dry run feed).
Reaction	Alarm display.
Remedy	Change the data to be entered before starting the part program.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

4240 Computation time overflow on the IPO or position controller level, IP %1

Explanation	%1 = program position The settings for the interpolation and position control clock have been modified prior to the last booting such that now too few computation time is available for the cyclic tasks to be performed. The alarm occurs immediately after booting if too few computing time is available for a task even with standing axes and the NC program not started. The task overflow, however, may also only occur if computation-time intensive NC functions are called during the program execution.
Reaction	NC not ready. NC Start inhibited. NC Stop at alarm. The NC switched to the follow-up mode. Alarm display. Interface signals are set.
Remedy	Please inform the authorized personnel/customer service. Optimize the clock time in NC MD 10050 **SYSCLOCK_CYCLE_TIME**, MD 10060 **POSCTRL_SYSCLOCK_TIME_RATIO** and/or MD 10070 **IPO_SYSCLOCK_TIME_RATIO** more cautiously. The test should be carried out using an NC program which constitutes a maximum possible load for the control system. For reasons of safety, the ties determined in this way should be provided with a safety reserve of 15 ... 25%.
Program continuation by	Power ON

4310

Illegal declaration in MD %1 index %2

Explanation

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

The values of the machine data must be written in the ascending sequence order.

Reaction

NC not ready.
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Correct the MD accordingly.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

4340

Channel %1 Invalid transformation type in transformation no. %2.

Explanation

%1 = channel number
%2 = transformation name

An invalid number, i.e. a number not defined was entered in one of the machine data TRAFO_TYPE_1 ... TRAFO_TYPE_8.
This alarm will also occur if a certain transformation type is not possible for the control system type specified.

Reaction

Alarm display
Interface signals are set
NC stop at alarm
Channel not ready
Mode group not ready
NC Start inhibited

Remedy

Enter a valid transformation type.

Program continuation by

Power ON

4343

Channel %1 It was tried to change the machine data of an active transformation.

Explanation

%1=channel number

It was tried to change the machine data of an active transformation in order to enable them via RESET or NEWCONFIG.

Reaction

Alarm display
Interface signals are set
NC Stop at alarm at the end of the block
Interpreter stop

Remedy

Set valid machine data.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

4346

Channel %1 Faulty geometry axis assignment in machine data %2[%3]

Explanation

%1=channel number
%2 = name of the machine data
%3 = transformation number

The machine data TRAFO_GEOAX_ASSIGN_TAB_1/2 contains an invalid entry.
The following error causes are possible:
- The entry refers to a channel axis not existing.

- The entry is zero (no axis) although the transformation requires the appropriate
  axis as a geometry axis.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm at the end of the block.

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

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

4347

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

Explanation
%1 = channel number
%2 = name of the machine data
%3 = transformation number

The machine data TRAFO_AXIS_IN_1/2 contains an invalid entry.
The following error causes are possible:
- The entry refers to a channel axis not existing.
- The entry is zero (no axis) although the transformation requires the appropriate
  axis as a channel axis.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm at the end of the block.

Remedy
Correct the entry in TRAFO_AXES_IN_1/2.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

4400

MD change results in reorganization of the battery-backed memory (loss of data!)

Explanation
An MD configuring the battery-backed memory was changed. Booting the NC with the changed data will reorganize the battery-backed memory, resulting in a loss of all battery-backed user data (part programs, tool data, GUD, LEC, ...).

Reaction
Alarm display.

Remedy
If the control system contains user data that have not been saved, a data backup must be carried out before the NC is next booted. The reorganization of the memory can be avoided by resetting the altered MD to the value during the last booting manually.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

4502

Channel %1 Anachronism: %2(%3) -> %4

Explanation
%1 = channel number
%2 = string: MD identifier
%3 = string: MD identifier
%4 = string: MD identifier

The reset behavior of the 6th or 8th G group has been defined in MD 20110 **RESET_MODE_MASK,** bit4 and bit5 to date; now, this setting is made in MD 20152 **GCODE_RESET_MODE**.

To be able to handle "old" data backups with full compatibility, the "old" values are taken from MD 20110 **RESET_MODE_MASK** and entered in MD 20152 **GCODE_RESET_MODE**.

Reaction: Alarm display.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

5000

Communication job cannot be executed

Explanation: The communication job (data exchange between NC and HMI, such as loading of an NC part program) cannot be carried out due to insufficient memory. Cause: Too many parallel communication jobs.

Reaction: Alarm display.

Remedy: No remedial action possible - the operator action that resulted in the alarm message must be repeated. Use "Cancel" to cancel the alarm.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6000

Memory mapping was carried out using the default machine data

Explanation: The memory management was not able to carry out the mapping of the NC user memory using the values in the machine data. The reason is that the entire memory is used by the NC user both as a dynamic memory and a static memory (e.g., for: number of tool offsets, number of directories and files, etc.).

Reaction: Alarm display.
Interface signals are set.
NC Start inhibited.
NC Stop at alarm.

Remedy: Redefine the memory mapping.

It is not possible to specify a certain MD as an alarm cause for the NC user memory mapping. Therefore, the MD that has caused the alarm must be determined on the base of the default values in the machine data and by modifying the user-specific memory mapping step by step.

In most cases, not only an individual MD is selected too large, and it is therefore recommended to reduce the memory area in several MDs correspondingly.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

6010

Channel %1 data block %2 was not or only partially created, error number %3

Explanation: %1 = channel number
%2 = string (block name)
%3 = internal error identifier

The data management has detected an error in the boot sequence. A possible reason is that the specified data block was not created. The type of the error can be derived from the error number. An error number >100000 indicates to a persisting system error. Otherwise, the user memory area has been dimensioned too small. In this case, the (user) error numbers have the following meaning:

Error number	Explanation
1	No more memory capacity
2	Number of maximum possible symbols exceeded
3	Index 1 outside the valid range of values
4	Name already exists in the channel
5	Name already exists in the NC

If the alarm occurs after loading cycle programs, macro definitions or definitions for global user data (GUD), the machine data for the user memory configuration have been dimensioned not correctly. In all the other cases, modifications to machine data that are already correct result in errors in the user memory configuration.

The following block names (2nd parameter) are known in the NC (both system and user data blocks in total; if problems occur only in the user data blocks, as a rule, these can be eliminated by user intervention)

_N_NC_OPT	- system-internal: Option data, NC-global
_N_NC_SEA	- system-internal: Setting data, NC global
_N_NC_SEA	- system-internal: Machine data, NC global
_N_NC_SEA	- system-internal: 'cross error compensation'
_N_NC_PRO	- system-internal: Protection areas, NC-global
_N_NC_GD1	- user: 1st GUD block defined by _N_SGUD_DEF, NC-global
_N_NC_GD2	- user: 2nd GUD block defined by _N_MGUD_DEF, NC-global
_N_NC_GD3	- user: 3rd GUD block defined by _N_UGUD_DEF, NC-global
_N_NC_GD4	- user: 4th GUD block defined by _N_GUD4_DEF, NC-global
_N_NC_GD5	- user: 5th GUD block defined by _N_GUD5_DEF, NC-globally
_N_NC_GD6	- user: 6th GUD block defined by _N_GUD6_DEF, NC-globally
_N_NC_GD7	- user: 7th GUD block defined by _N_GUD7_DEF, NC-globally
_N_NC_GD8	- user: 8th GUD block defined by _N_GUD8_DEF, NC-globally
_N_NC_GD9	- user: 9th GUD block defined by _N_GUD9_DEF, NC-globally
_N_NC_MAC	- user: Macro definitions
_N_NC_FUN	- user: Cycle programs

_N_CHc_OPT - system-internally: Option data, channel-specific
_N_CHc_SEA - system-internal: Setting data, channel-specific
_N_CHc_SEA - system-internal: Machine data, channel-specific
_N_CHc_SEA - system-internal: Protection areas, channel-specific
_N_CHc_SEA - 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_SEA - system-internal: Machine data, axial
_N_AXa_EEC - system-internal: Leadscrew error compensation data, axial
_N_AXa_QEC - system-internal: Quadrant error compensation data, axial

_N_TOt_TOC - system-internal: Toolholder data, TOA-specific
_N_TOt_TOA - system-internal: Tool data, TOA-specific
_N_TOt_TMA - system-internal: Magazine data, TOA-specific

c = channel number
a = machine axis number
t = TOA unit number

Reaction

There are further internal system data blocks with identifiers.

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.
NC Stop at alarm.

Remedy

Either correct the machine data or undo the changes.

Please inform the authorized personnel/customer service.

- There are two decisive machine data for cycle programs:
  MD 18170 **MM_NUM_MAX_FUNC_NAMES** = max. number of all cycle programs
  Error number = 2 indicates that the value concerned is too small
  MD 18180 **MM_NUM_MAX_FUNC_PARAM** = max. number of all parameters defined in the cycle programs
  Error number = 2 indicates that the value concerned is too small

  (if this MD is changed, the data contained in the battery-backed memory remain stored)

- The following applies to macro definitions:
  MD 18160 **MM_NUM_USER_MACROS** = max. number of all macro definitions
  Error number = 2 indicates that the value concerned is too small

(if this MD is changed, the data contained in the battery-backed memory remain stored)

- The following applies to GUD variables:
  MD 18118 **MM_NUM_GUD_MODULES** = max. number of GUD data blocks per area (NC/channel)
  (if GD1, GD2, GD3, GD9 are to be defined, the value must be =9 and not =4)
  MD 18120 **MM_NUM_GUD_NAMES_NCK** = max. number of all NC-global GUD variables
  Error number = 2 indicates that the value concerned is too small
  MD 18130 **MM_NUM_GUD_NAMES_CHAN** = max. number of all channel-specific GUD variables in the channel
  Error number = 2 indicates that the value concerned is too small
  MD 18150 **MM_GUD_VALUES_MEM** = entire value memory of all GUD variables together
  Error number = 1 indicates that the value concerned is too small

Power ON

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

6020 Machine data changed - memory remapped

Explanation: Machine data defining the NC user memory have been changed. The data management has remapped the memory according to the changed machine data.

Reaction: Alarm display.

Remedy: No remedial action required. Re-enter the required user data.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

6030 User memory limit has been adapted

Explanation: When booting, the data management checks the really existing, physical user memory (DRAM, DPRAM and SRAM) against the values in the system-specific machine data MD 18210 **MM_USER_MEM_DYNAMIC**, MD 18220 **MM_USER_MEM_DPR** and MD 18230 **MM_USERMEM_BUFFERED.**

Reaction: Alarm display.

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

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

6035 System has only %2 kB free user memory of the type "%3" instead of only %1 kB

Explanation: %1 = free memory capacity in kB defined for the control system model concerned
%2 = real maximum capacity of free memory in kB
%3 = memory type, "D" = not battery-backed, "S" = battery-backed

This alarm may only occur after a 'cold restart' (= NCK booting with the default machine data). The alarm is only a note; the NCK functions are not affected. This alarm indicates that the NCK has fewer user memory than intended by Siemens for this control system variant. The value of the really free user memory can also be taken from the machine data $MN_INFO_FREE_MEM_DYNAMIC, $MN_INFO_FREE_MEMS_TATIC.

Siemens delivers the NCK with default settings which depending on the particular model provide a certain (free) memory for the specific settings of the particular applications. Original NCK systems are set by default such that this alarm does not occur in cold restart.

Reaction: Alarm display.

Remedy:
- This message may have the following possible causes:
- The NCK is run on a hardware that is not intended for this NCK release (in other words, that has insufficient memory capacity).
- If the particular application can manage with the remaining free user memory (in other words, can be started up without errors), the message can also simply be ignored.

Program continuation by: Use the RESET key to cancel the alarm.

6410

TO unit %1 tool "%2" / duplo no. %3 has reached prewarning limit with D= %4

Explanation:
%1 = TO unit
%2 = tool identifier (name)
%3 = duplo number
%4 = D number

Tool monitoring:
The operator is referred to the fact the specified D compensation of the tool monitored with reference to the time or count has reached its prewarning limit.
If possible, the D number is specified; if not, the 4th parameter will be 0.

The particular type of the tool monitoring is a property of the tool (see $TC_TP9).

If you do not work with replacement tools,
the specification of the duplo number has no further meaning.

The alarm is triggered either via MMC or PLC (= OPI). The channel context is not defined. For this reason, the TO unit is specified.

Reaction: Alarm display.
Interface signals are set.

Remedy: only intended for information; the user will decide what to do.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6411

Channel %1 tool "%2" / duplo no. %3 has reached prewarning limit with D= %4

Explanation:
%1 = TO unit
%2 = tool identifier (name)
%3 = duplo number
%4 = D number

Tool monitoring:
The operator is referred to the fact the specified D compensation of the tool monitored with reference to the time or count has reached its prewarning limit.
If possible, the D number is specified; if not, the 4th parameter will be 0.

The particular type of the tool monitoring is a property of the tool (see $TC_TP9).

If you do not work with replacement tools, the specification of the duplo number has no further meaning.
This alarm is caused within the framework of the NC program execution .

Reaction	Alarm display. Interface signals are set.
Remedy	only intended for information; the user will decide what to do.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6412

TO unit %1 tool "%2" / duplo no. %3 has reached monitoring limit with D= %4

Explanation	%1 = TO unit %2 = tool identifier (name) %3 = duplo number %4 = D number Tool monitoring: The operator is referred to the fact the specified D compensation of the tool monitored with reference to the time or count has reached its monitoring limit. If possible, the D number is specified; if not, the 4th parameter will be 0. The particular type of the tool monitoring is a property of the tool (see $TC_TP9). If you do not work with replacement tools, the specification of the duplo number has no further meaning. The alarm is triggered either via MMC or PLC (= OPI). The channel context is not defined. For this reason, the TO unit is specified.
Reaction	Alarm display. Interface signals are set.
Remedy	only intended for information; the user will decide what to do.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6413

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

Explanation	%1 = TO unit %2 = tool identifier (name) %3 = duplo number %4 = D number Tool monitoring: The operator is referred to the fact the specified D compensation of the tool monitored with reference to the time or count has reached its monitoring limit. If possible, the D number is specified; if not, the 4th parameter will be 0. The particular type of the tool monitoring is a property of the tool (see $TC_TP9). If you do not work with replacement tools, the specification of the duplo number has no further meaning. This alarm is caused within the framework of the NC program execution.
Reaction	Alarm display. Interface signals are set.
Remedy	only intended for information; the user will decide what to do.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6430 Workpiece counter: Overflow of table of monitored cutting edges

Explanation

No further cutting edges can be entered in the workpiece counter table.
You can store as many cutting edges for the workpiece counter as totally possible in the NCK.
In other words: If each cutting edge of every tool is used for a workpiece exactly once, the limit is reached.
If at the same time several workpieces are machined at several toolholders/spindles, cutting edges can be stored for the workpiece counter for the whole range of workpieces using
MD18100 MM_NUM_CUTTING_EDGES_IN_TOA.
If the alarm occurs, this means that cutting edges which will be used as of now are no longer monitored for their count; this applies until the table is emptied, e.g. via the NC language command SETPIECE or the relevant job from the MMC, PLC (PI service).

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

- Decrementing workpiece counter forgotten?
  If so, program SETPIECE in the part program or include the appropriate command in the PLC program.
- If the part program or the PLC program is correct, more memory should be freed for the cutting edges using the machine data
  $MN_MM_NUM_CUTTING_EDGES_IN_TOA (this can only be done by persons who have the appropriate access rights).

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

6431 Illegal function; tool management / monitoring not activated

Explanation

A function of the data management has been called which is not available due to the tool management or tool monitoring not enabled; this may pertain, e.g. to the language commands GETT, SETPIECE, GETSELT, NEWT, DELT.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

- Please inform the authorized personnel/customer service.
- Check the desired configuration of the CNC. Are tool management or tool monitoring necessary, but not activated?
- Is a part program used what is designed for a CNC with tool management/ tool monitoring? And even this program is started on a CNC not providing tool management/tool monitoring. This is not possible! Either run the part program on an appropriate CNC or modify the part program.
- Activate the tool management/tool monitoring by setting the appropriate machine data; see $MN_MM_TOOL_MANAGEMENT_MASK, $MC_TOOL_MANAGEMENT_MASK.
- Check whether the required option is set.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

6432 Function cannot be executed; no tool is mounted on the spindle; channel %1

Explanation: %1=channel number

You have probably tried to carry out an operation in the part program of the appropriate channel, which requires a tool mounted on the toolholder/spindle. This can be, e.g. the workpiece count monitoring function.

Reaction: Alarm display.
Interface signals are set.

Remedy:
- Select a different function, a different toolholder/spindle or mount a tool on the toolholder/spindle.

Program continuation by: Use either the "Cancel" key or press NC START to cancel the alarm.

6500 NC memory limit reached

Explanation: Too many programs have been loaded. The job cannot be executed.

During the commissioning, this may pertain to files of the NC file system (part of the NC memory), such as initialization files, NC programs etc.

Reaction: Alarm display.

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

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6510 Too many part programs in NC memory

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

Reaction: Alarm display.

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

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6530 Too many files in directory

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

Reaction: Alarm display.

Remedy: Delete or unload files (e.g. part programs) from the directory concerned.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6540 Too many directories in NC memory

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

Reaction: Alarm display.

Remedy: Delete or unload directories not needed (e.g. workpiece).

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

6550 Too many subdirectories

Explanation	The number of subdirectories in a directory of the NC has reached the maximum.
Reaction	Alarm display.
Remedy	Please inform the authorized personnel/customer service. Delete or unload subdirectories from the directory concerned.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6560 Illegal data format

Explanation	Illegal data are entered in a file of the NC; this may occur, e.g. if binary data are loaded into the NC as an ASCII file.
Reaction	Alarm display.
Remedy	Mark the file as binary data (e.g. via the extension .BIN)
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6570 NC memory limit reached

Explanation	The DRAM file system of the NC is full. The job cannot be executed. Too many system files have been created in the DRAM.
Reaction	Alarm display.
Remedy	Start fewer "Program execution from an external source" processes.
Program continuation by	

6600 NC card memory limit reached

Explanation	The NC card file system of the NC is full. No further data can be stored on the NC card.
Reaction	Alarm display.
Remedy	Delete data from the PCMCIA card.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6610 Too many files opened on the NC card

Explanation	Too many files on the NC card are accessed at the same time.
Reaction	Alarm display.
Remedy	Repeat the action later.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6620 Invalid NC card format

Explanation	The NC card cannot be accessed because of the invalid NC card format.
Reaction	Alarm display.
Remedy	Replace the NC card.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6630 NC card hardware defective

Explanation	The NC card cannot be accessed because the card is defective.
Reaction	Alarm display.
Remedy	Replace the PCMCIA card.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6640 NC card not plugged

Explanation	The NC card cannot be accessed because no card is inserted.
Reaction	Alarm display.
Remedy	Insert the NC card.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6650 Write protection of NC card active

Explanation	Writing to the NC card is not possible since the write protection is active.
Reaction	Alarm display.
Remedy	Deactivate the write protection.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6660 Flash File System option not set

Explanation	Writing to the NC card is not possible since the relevant option is not set.
Reaction	Alarm display.
Remedy	Buy the option.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

6670 Data are being read from NC card

Explanation	This alarm is issued while the contents of the NC card are being read. During this time, the FFS cannot be accessed.
Reaction	Alarm display.
Remedy	Wait until reading is finished.
Program continuation by	The alarm display will disappear with the alarm cause. No further operation required.

6671 Data are being written to NC card

Explanation	This alarm is issued while the contents of the NC card are being written. During this time, the flash file system (FFS) cannot be accessed. If POWER OFF is provided during the time when the alarm exists, the contents of the NC card are destroyed!
Reaction	Alarm display.
Remedy	Wait until writing is finished.
Program continuation by	The alarm display will disappear with the alarm cause. No further operation required.

6693

File %1 is lost

Explanation: %1 = file name

Reaction: The file change could not correctly be completed due to power failure. The file is lost.

Alarm display.
Interface signals are set.
NC not ready.
NC Start inhibited.

Remedy: Reload the file.

Program continuation by: Power ON

6698

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

Explanation: Writing to the NC card is not possible since no valid writing algorithm exists for the flash memory.

Reaction: Alarm display.

Remedy: Either install a compatible NC card or, after consultation of SIEMENS, enter the new manufacturer code/device code in MD 11700 **PERMISSIVE_FLASH_TAB.**

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

8040

MD %1 reset; appropriate option not set

Explanation: %1 = string: MD identifier

A machine data was set which is interlocked via an option.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/customer service.
Please contact your machine manufacturer or a sales representative from A&D MC of SIEMENS AG for ordering the required option.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

8041

Axis %1: MD %2 reset; appropriate option not sufficient

Explanation: %1 = axis number
%2 = string: MD identifier

The number of axes selected in the relevant option machine data is exhausted. In the specified machine axial machine data, functions for too many functions belonging to the option have been selected.

Reaction: Alarm display.
Interface signals are set.
NC Start inhibited.
NC Stop at alarm.
NC not ready.
Under certain circumstances, it can be switched across all channels via MD.

Program continuation by: Power ON

10203

Channel %1 NC start not possible without reference point

Explanation: %1=channel number

NC START has been pressed either in MDA or AUTOMATIC mode, and at least one axis to be referenced has not reached its reference point.

Reaction: Alarm display.

Remedy: Referencing start can be initiated either channel or axis-specifically.

1. **Channel-specific reference-point approach:** The rising edge of the interface signal "Activate referencing" (V 32000001.0) starts an automatic cycle that starts the axes of the channel in the order specified in the axis-specific MD 34 110 **REFP_CYCLE_NR** (axis order for channel-specific referencing).
   - **-1:** Axis is not involved in channel-specific referencing, but has to be referenced for NC start
   - **0:** Axis is not involved in channel-specific referencing, but must be referenced for NC start.
   - **1-4:** Starting order for channel-specific referencing (axes/channels with the same number are started simultaneously)
2. **Axis-specific referencing:** Press the direction key corresponding to the approach direction specified in the axis-specific MD 34 010 REFP_**CAM_DIR_IS**_MINUS (Approach reference point in the minus direction).

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

10208

Channel %1 Press NC START to continue the program

Explanation: %1=channel number

After block search with calculation, the control system is in the desired state. The program can now be started with NC Start or in the state first changed by Overwrite/Jog.

Reaction: Alarm display.
NC Stop at alarm.

Remedy: Press NC START.

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

10225

Channel %1:Command % 2 denied

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

The channel has received a command which cannot be executed.

Reaction: Alarm display.

Remedy: Press RESET.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

10299

Channel %1 Auto Repos function not enabled

Explanation: `%1 = channel number`

The Auto Repos function (mode) which is not implemented was selected in the channel.

Reaction: Alarm display.

Remedy: This is only a note.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

10601 Channel %1 block %2 End-of-block velocity when thread cutting is zero

Explanation

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

This alarm occurs only if several blocks with G33 follow one after another. The end-of-block velocity in the specified block is zero although still one more velocity block follows. Possible causes are, for example:

- G09
- Auxiliary function after a motion
- Output of auxiliary function prior to the motion of the subsequent block

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Modify the NC part program (do not program "Stopping at end of block" G09).

Change the general machine data 11110 **AUXFU_GROUP_SPEC [n]** for selecting the output time of an auxiliary function group from "Auxiliary function output prior to/after the movement" to "Auxiliary function output during the motion".

Bit 5 = 1: Auxiliary function output **prior** to the motion
Bit 6 = 1: Auxiliary function output **during** to the motion
Bit 7 = 1: Auxiliary function output **after** the motion

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10604 Channel %1 block %2 Thread pitch increase too high

Explanation

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

The increase in the thread pitch results in an axis overload.
The check assumes a spindle override of 100%.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy

Reduce spindle speed, thread pitch increase or path length in the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10605 Channel %1 block %2 Thread pitch reduction too high

Explanation

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

The reduction in the thread pitch results in axis standstill in the thread block.

Reaction

Alarm display
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy

Reduce the increase in the thread pitch or the path length in the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10607 Channel %1 block %2 Thread cannot be executed with frame

Explanation

%1 = channel number
%2 = Block number, label

The current frame falsifies the reference between thread length and thread pitch.

Reaction

Alarm display.
Interface signals are set.
NC Stop at alarm at block end.
NC Start inhibited.

Remedy

Carry out thread cutting with G33, G34, G35 without frame.
Use G63 or G331/G332.

Program continuation by

Press the RESET key to cancel the alarm. Restart the part program.

10620 Channel %1 block %3 axis %2 reaches software limit switch %4

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = block number, label
%4 = string

During the traversing movement, it is detected that the software limit switch has been overtraveled in the displayed direction. The overshooting of the traversing range could not yet be detected in the block preparation, since a motion overlay by the **handwheel** takes place.

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

Depending on the cause that initiated the alarm, use the following remedy:

1. Handwheel override: Undo the motion overlay and avoid /reduce it when repeating the program.
2. Transformation: Check the set/programmed zero offset (current frame). If the values are correct, offset the tool mounting device to avoid that always the same alarm is generated (with recurring abnormal program termination).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10621 Channel %1 axis %2 rests on software limit switch %3

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = string

The specified axis is already on the displayed software limit position.

Reaction

Alarm display.

Remedy

Check the machine data
MD 36110 **POS_LIMIT_PLUS/**MD 36130 **POS_LIMIT_PLUS2** and MD 36100 **POS_LIMIT_MINUS/**MD 36120 **POS_LIMIT_MINUS2**
for the software limit switches.

Check the axis-specific interface signals "2nd software limit switch plus" (V 380x1000.3) and "2nd software limit switch minus" (V 380x1000.2) to make sure that the 2nd software limit switch is selected.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

10630

Channel %1 block %2 axis %3 reaches working area limitation %4

Explanation

%1 = channel number
%4 = axis name, spindle number
%2 = block number, label
%3 = string

The specified axis violates the working area limitation. This is only detected in the main run, since either the minimum axis values could not be determined prior to the transformation or an overlaid movement is present.

Reaction

Alarm display.
Interface signals are set.
NC Stop at alarm at block end.
NC Start inhibited.

Remedy

Program a different motion or do not carry out an overlaid motion.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10631

Channel %1 axis %2 stands on working area limitation %3

Explanation

%1 = channel number
%2 = axis, spindle
%3 = string (+ or -)

In the JOG mode, the specified axis reaches the positive working area limitation.

Reaction

Alarm display.

Remedy

Check the setting data
SD 43420 **WORKAREA_LIMIT_PLUS** and
SD 43430 WORKAREA_LIMIT_MINUS for the working area limitation.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

10720

Channel %1 block %3 axis %2 software limit switch %4

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = block number, label
%4 = string (+ or -)

The path programmed for the axis violates the software limit switch currently active. (The 2nd 2nd software limit switch becomes active with the interface signal "2nd software limit switch plus/minus" in V 380x1000.and .3). The alarm is activated when preparing the part program block.

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

Check the **position** specified for the axis in the part program.

Check the machine data
MD 16100 **POS_LIMIT_MINUS**/MD 36120 **POS_LIMIT_MINUS2** and
MD 36110 **POS_LIMIT_PLUS**/ MD 36130 POS_**LIMIT_PLUS2**
for the software limit switches.

Check the axis-specific **interface signals** "2nd software limit switch minus" (V 380x1000.23) to make sure that the 2nd software limit switch is selected.

Check the currently active **zero offsets** via the current frame.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10730 Channel %1 block %3 axis %2 Working area limitation %4

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = block number, label
%4 = string (+ or -)

If you detect during the block preparation that the programmed path will lead the axis across the working area limitation, this alarm is generated.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy

a) Check the **NC program** for correct position data.

b) **Check the zero offsets** (current frame).

c) Correct the **working area limitation** with G25 or

d) Correct the **working area limitation** via setting data or

e) Deactivate the **working area limitation** via SD 43410 **WORKAREA_MINUS_ENABLE=FALSE.**

Program continuation by

Press NC START to cancel the alarm and continue the program.

10740 Channel %1 block %2 Too many empty blocks in SAR programming

Explanation

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

No more than 5 blocks may be programmed between the SAR block and the block determining the approach or retraction tangent.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Modify the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10741 Channel %1 block %2 Direction reversal in SAR infeed motion

Explanation

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

A safety clearance has been programmed whose direction vertically to the machining plane lies not between the start and end points of the SAR contour.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Modify the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10742

Channel %1 block %2 SAR distance invalid or not programmed

Explanation

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

Possible causes are:
The parameter DISR was not specified in an SAR block or its value is less than or equal to 0.

When approaching or retracting along a circle and the tool radius compensation active, the radius of the SAR contour internally created is negative. The SAR contour internally created is a circle with a radius - when corrected using the current correction radius (total of tool radius and OFFN offset value), results in the tool center point path with the programmed radius DISR.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Modify the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10743

Channel %1 block %2 SAR programmed repeatedly

Explanation

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

You had tried to activate an SAR motion before an SAR motion first activated was completed.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Modify the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10744

Channel %1 block %2 No valid SAR direction defined

Explanation

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

The tangent direction is not defined with smooth approach / retraction (SAR).

Possible causes:
The approach block in the program is not followed by a block with traversing information.

No block with traversing information was programmed in the program prior to a traversing block.

The tangent to be used for the SAR motion stands vertically to the current machining plane.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Modify the part program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10745

Channel %1 block %2 SAR end position not unambiguous

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

Both in the SAR block and in the subsequent block, the position was programmed vertically to the machining direction, and in the SAR block, no position was specified for the machining plane.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy: Modify the part program.
Either remove the position specification for the infeed axis from the SAR block or from the subsequent block, or program a position in the machining plane also in the SAR block.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10746

Channel %1 block %2 Preprocessing stop with SAR

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

A preprocessing stop was inserted between an SAR block and the subsequent block defining the tangent direction or between an SAR retraction block and the subsequent block defining the end position.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy: Modify the part program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10747

Channel %1 block %2 Retraction direction not defined with SAR

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

In an SAR retraction block with quadrant or semi-circle (G248 or G348), the end point in the machining plane was not programmed, and either G143 or G140 is active without tool radius compensation.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy: Modify the part program.
The following changes are possible:

- Specify the end point in the SAR block for the machining plane.
- Activate the tool radius compensation (only effective with G140, not with G143).

- Specify the retraction side explicitly with G141 or G142.
- Retract along a circle instead of a straight line.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10750

Channel %1 block %2 tool radius compensation activated without tool no.

Explanation:

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

A tool T.. must be selected so that the control system can take into account the appropriate compensation values.

Each tool (T number) is automatically assigned a compensation data block (D1) containing the compensation data. A tool can be assigned max. 9 compensation data blocks by specifying the desired data block with the D number. (D1 - D9).

The tool radius compensation (TRC) will be taken into account if the function G41 or G42 is programmed. The compensation values are to be found in the parameter P6 (geometry value) and P15 (wear value) of the active compensation data block D_X.

Reaction:

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy: Before calling the TRC using G41/G42, program a tool no. under the address T....

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

10751

Channel %1 block %2 danger of collision in tool radius compensation

Explanation:

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

The 'bottle neck detection' (calculation of the intersection point of the following, corrected traversing blocks) was not able to calculate an intersection point for the overviewed number of traversing blocks, resulting in the risk that one of the equidistant paths will violate the workpiece contour.

Reaction:

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy: Check the part program and modify the program (if possible) such that inner corners with paths shorter than the compensation value are avoided. (Outer corners are not critical, since the equidistants are extended or intermediate blocks are inserted so that an intersection point is always provided).

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

10752 Channel %1 block %2 Overflow of local block buffer in tool radius compensation

Explanation

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

The tool radius compensation must buffer an alternating number of intermediate blocks in order to be able to calculate the equidistant tool path for each NC block. The size of the buffer memory cannot easily be determined. It depends on the number of blocks without traversing information in the compensation level and the number of contour elements to be inserted.

The size of the buffer memory is fixed by the system and cannot be altered via MD.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Reduce the buffer memory which has been occupied by modifying the NC program. This can be achieved if you avoid the following:

- blocks without traversing information in the compensation plane
- blocks that contain contour elements with variable curves (e.g. ellipses) and curve radii which are smaller than the compensation radius. (Such blocks are split into several partial blocks).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10753 Channel %1 block %2 Tool radius compensation can only be selected in a linear block

Explanation

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

Tool radius compensation with G41/G42 may only be selected in blocks with the G function G00 (rapid traverse) or G01 (feed) active.
At least one axis of the plane G17 to G19 must be written in the block that contains G41/G42; it is always recommended to program both axes, since usually also both axes traverse when selecting the compensation.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Correct the NC program; replace the compensation selection with a block with linear interpolation.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10754

Channel %1 block %2 Tool radius compensation can only be deselected in a linear block

Explanation
%1 = channel number
%2 = block number, label

The deselection of the tool radius compensation by G40 may only be carried out in blocks in which the G function G00 (rapid traverse) or G01 (feed) is active.
At least one axis of the plane G17 to G19 must be written in the block that contains G40; it is always recommended to program both axes, since usually also both axes traverse when deselecting the compensation.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
Correct the NC program; relocate the compensation deselection to a block with linear interpolation.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10755

Channel %1 block %2 Tool radius compensation cannot be enabled in current starting point using KONT

Explanation
%1 = channel number
%2 = block number, label

When activating the cutter radius compensation via **KONT**, the starting point of the approach block is within the compensation circle, thus already violating the contour.

If the cutter radius compensation is selected using G41/G42, the approach behavior (NORM or KONT) will determine the compensation motion if the current actual position is **after** the contour. If KONT is active, a circle with the cutter radius is drawn around the programmed starting point (= end point of the approach block). The tangent that leads through the current actual position and that does not violate the contour is the approach movement.

If the starting point is within the compensation circle and around the target point, the tangent will lead through this point.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
Place the selection of the cutter radius compensation such that the starting point of the approach movement will be outside the compensation circle around the target point (programmed traversing movement > compensation radius). The following possibilities are provided:

- Selection in the previous block
- Insert intermediate block
- Select approach behavior NORM

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10756 Channel %1 block %2 Tool radius compensation cannot be disabled at current starting point using KONT

Explanation

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

When deselecting the cutter radius compensation, the programmed end point is within the compensation circle. If this point would really be approached without compensation, a contour violation would be the consequence.

If the cutter radius compensation is disabled using G40, the starting behavior (NORM or KONT) will determine the compensation movement if the programmed end point is **after** the contour. If KONT is active, a circle with the cutter radius is drawn around the point at which the compensation is still active. The tangent that leads through the programmed end position and that does not violate the contour is the starting movement.

If the programmed end point is within the compensation circle around the target point, no tangent will lead through this point.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Place the deselection of the cutter radius compensation such that the programmed end point is outside the compensation circle around the last active compensation point. The following possibilities are provided:

- Deselection in the next block
- Insert intermediate block
- Select starting behavior NORM

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10758 Channel %1 block %2 Bend radius with variable compensation value too small

Explanation

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

The current cutter radius compensation (the used cutter) is too large for the programmed path radius.

In a block with variable tool radius compensation, a compensation must be possible either in any or none place of the contour from the programmed area using the smallest **and** the largest compensation value. No point on the contour is admitted, in which the bend radius is within the variable compensation area.

If the compensation value changes its sign within a block, both sides of the contour will be checked; otherwise, only the compensation side.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy

Either use a smaller cutter or take into account a part of the cutter radius already in the contour programming.

Program continuation by

Press NC START to cancel the alarm and continue the program.

10762

Channel %1 block %2 Too many empty blocks between two traversing blocks with active tool radius compensation

Explanation
%1 = channel number
%2 = block number, label

The maximum permissible number of empty blocks is limited

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
1. Modify the part program:
2. Check whether SBL2 is selected. When SBL2 is active, a block from each part program line is generated resulting in that the permissible number of empty blocks between two traversing blocks can be exceeded.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10763

Channel %1 block %2 Path component of block in compensation plane becomes zero.

Explanation
%1 = channel number
%2 = block number, label

Because of the collision monitoring with tool radius compensation, the path component of the block in the compensation plane becomes zero. If no information with regard to the motion perpendicular to the compensation plane is contained in the original block, the block is skipped.

Reaction
Alarm display.

Remedy
- The behavior at narrow places which cannot be machined with the active tool is correct.
- Modify the part program.
- If necessary use a tool with a smaller radius.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

10764

Channel %1 block %2 Discontinuous path with active tool radius compensation

Explanation
%1 = channel number
%2 = block number, label

This alarm occurs if the tool radius compensation is active and the starting point used for the calculation of the compensation is not equal to the end point of the previous block.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
Modify the part program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10776 Channel %1 block %2 axis %3 must be a geometry axis with tool radius compensation

Explanation
%1 = channel number
%2 = block number, label
%3 = axis name

This alarm occurs if an axis for which a tool radius compensation is required is not a geometry axis.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10777 Channel %1 block %2 Tool radius compensation:Too many blocks with suppression of compensation

Explanation
%1 = channel number
%2 = block number, label

The maximum permissible number of blocks with active compensation suppression with tool radius compensation active is limited.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
- Modify the part program.
- Check whether SBL2 is selected. When SBL2 is active, a block from each part program line is generated resulting in that the permissible number of empty blocks between two traversing blocks can be exceeded.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

10778 Channel %1 block %2 Preprocessing stop with active tool radius compensation

Explanation
%1 = channel number
%2 = block number, label

If preprocessing stop is detected with the tool radius compensation active (either programmed by the user or generated internally), this alarm is generated, since in this situation machine movements might occur which are not intended by the user (completing radius compensation and re-approaching the contour). Program execution can be continued by pressing CANCEL and restart.

Reaction
Alarm display.

Remedy
- Press CANCEL and START to continue.
- Modify the part program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10780 Channel %1 block %2 Preprocessing stop with active tool radius compensation

Explanation
%1 = channel number
%2 = block number, label

If preprocessing stop is detected with the tool radius compensation active (either programmed by the user or generated internally), this alarm is generated, since in this situation machine movements might occur which are not intended by the user (completing radius compensation and re-approaching the contour).

Press CANCEL and restart the system to continue.

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

Remedy
- Press CANCEL and START to continue.
- Modify the part program.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

10790 Channel %1 block %2 Plane change when programming straight lines with angles

Explanation
%1 = channel number
%2 = block number, label

When programming two straight lines with angle specification, you have changed the active plane between the first and the second partial blocks.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10791 Channel %1 block %2 Invalid angles when programming straight lines

Explanation
%1 = channel number
%2 = block number, label

When programming a contour definition consisting of two straight lines and specifying angles, no intermediate point could be found.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10792 Channel %1 block %2 Illegal interpolation type when programming straight lines with angles

Explanation
%1 = channel number
%2 = block number, label

When programming two straight lines with specifying angles, only spline or linear interpolation is permitted. Circular or polynomial interpolation are not allowed.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10793 Channel %1 block %2 Second block of linear interpolation with angles missing

Explanation
%1 = channel number
%2 = block number, label

When programming two straight lines with specifying the angles, the second block is missing. This case occurs if the first partial block is at the same time the last block of a program or if the first partial block is followed by a block with preprocessing stop.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10794 Channel %1 block %2 Angle specification is missing in straight interpolation with angles

Explanation
%1 = channel number
%2 = block number, label

When programming two straight lines with specifying the angles, the angle specification in the second block is missing.
This error may only occur if in the previous block an angle, but not an axis of the active plane has been programmed.
Another possible error cause may be that it was intended to program an individual straight line with angle in the previous block. In this case, (exactly) one axis of the active plane must be programmed in this block.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy
Modify the part program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10795

Channel %1 block %2 End point specification in angle programming inconsistent

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

When programming a straight line, both positions of the active plane and an angle have been specified (the position of the end point is thus overdetermined), or the position of the programmed coordinate cannot be reached with the specified angle.
To program a contour definition consisting of two straight lines using angles, this specification of two axis positions in the plane and of an angle in the second block is admissible. The error can therefore also occur if the previous block could not be interpreted as the first partial block of such a contour definition due to a programming error.
In this case, a block will be interpreted as a the first block of a contour definition, which consists of two blocks, if an angle, but not an axis of the active plane has been programmed and if the block concerned is not already the second block of a contour definition.

Reaction: Alarm display.
Interface signals are set.
NC Stop at alarm.
Under certain circumstances, it can be switched over for several channels via MD.
The NC will switch to the follow-up mode.

Remedy: Modify the part program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10810

Channel %1 block %2 Master spindle not defined

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

The function "Revolutional feedrate" (with G95 or G96) or "Rigid tapping" (with G331/G332) was programmed although no master spindle is defined from which the speed could be taken.

The MD 20090 **SPIND_DEF_MASTER_SPIND** for the default setting or the vocabulary word **SETMS** are provided for the definition in the part program so that each spindle of the channel can be redefined to a master spindle.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Set the master spindle to an appropriate default value using MD 20090 **SPIND_DEF_MASTER_SPIND[n]=m** (n ... channel index, m ... spindle no.) or define it in the NC program using an appropriate identifier before a G function is programmed which requires a master spindle.

The machine axis which is to be operated as a spindle must be assigned a spindle number specified in
MD 35000 **SPIND_ASSIGN_TO_MACHAX[n]=m** (n ... machine axis index, m ... spindle no.). In addition, it must be assigned to a channel (channel axis index 1 or 2) via the MD 20070 **AXCONF_MACHAX_USED[n]=m** (n ... channel axis index, m ... machine axis index).

Program continuation by: Press NC START to cancel the alarm and continue the program.

10820

Channel %1 No rotary axis/spindle %2 defined

Explanation: %1 = channel number
%2 = axis name, spindle number

A revolutional feedrate was programmed for path and synchronous axes or for an axis/spindle, but the rotary axis from which the feedrate is to be derived is not available.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Correct the part program or set setting data 43300 **ASSIGN_FEED_PER_REV_SOURCE** correctly.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10860

Channel %1 block %2 No feedrate programmed

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

An interpolation type other than G00 (rapid traverse) is active in the displayed block. No F value programmed.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Program the feedrate value with reference to the interpolation type used.

- **G93:** The feedrate is specified under **F** as a time-reciproke value in [1/min].
- **G94 a**nd **G97:** The feedrate is programmed under address **F** in [mm/min] or [m/min].
- **G95:** The feedrate is programmed as a revolutional feedrate in [mm/rev] under address **F**.
- **G96:** The feedrate is programmed in [m/min] as a cutting speed in [mm/rev] under address **S**.m It results from the current **spindle speed**.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

10862

Channel %1 block %2 Master spindle also used as a path axis

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

A path has been programmed, which also contains master spindle as a path axis.
The path velocity, however, is derived from the master spindle speed (e.g. G95).

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Modify the program such that no reference to itself is possible.

Program continuation by
Press NC START to cancel the alarm and continue the program.

10870

Channel %1 block %2 No transverse axis defined

Explanation
%1 = channel number
%2 = block number, label

When the constant cutting speed is selected using the function **G96**, the spindle speed is controlled via the position of the transverse axis such that the cutting speed programmed under S [mm/min] will result at the tool edge.

In MD 20100 **DIAMETER_AX_DEF[n,m]=**x (n .. channel index, m .. spindle index, x .. axis name), the name of the transverse axis [string] can be specified for each of the 5 spindles used for the speed calculation.20100

$$S\ [1/\min] = \frac{S_{G96}\,[\mathrm{m/min}] \cdot 1000}{D_{\mathrm{Planachse}}\,[\mathrm{mm}] \cdot \pi}$$

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy
Enter the name of the transverse axis in **MD 20100 DIAMETER_AX_DEF** for the spindles used.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10880

Channel %1 block %2 Too many empty blocks between two traversing blocks when inserting chamfers or radii

Explanation
%1 = channel number
%2 = block number, label

Too many blocks without contour information are programmed between two blocks which contain contour elements and which are to be linked with a chamfer or a radius **(CHF, RND)**.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy
Modify the part program such that the permissible number of empty blocks is not exceeded.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10881

Channel %1 block %2 Overflow of local block buffer when inserting chamfers or radii

Explanation
%1 = channel number
%2 = block number, label

The number of empty blocks without contour information between two blocks which contain contour elements and which are to be linked with a chamfer or radius **(CHF, RND)** is so large that the internal buffer memory is too small.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy
Modify the part program such that the number of empty blocks is reduced.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10882

Channel %1 block %2 Activation of chamfers or radii (not modally) without traversing motion in the block

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

No chamfer or radius has been inserted between 2 linear or circle contours (edge breaking), because:

- no straight line or circle contour exists in the plane
- a motion exists outside the plane
- the plane has been changed
- the permissible number of empty blocks without traversing information (dummy blocks) has been exceeded

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Correct the part program, taking into account the error mentioned above.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

10883

Channel %1 block %2 Chamfer or radius must be reduced.

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

This alarm is generated if - when inserting chamfers or radii of at least one of the blocks involved is so short that the contour element to be inserted must be reduced with reference to the value actually programmed. This alarm occurs only if bit 4 is set in the machine data $MN_ENABLE_ALARM_MASK. Otherwise, the chamfer or rounding is adapted with generating an alarm.

Reaction: Alarm display.
Interface signals are set.
NC Stop at alarm at end of block.

Remedy: Modify the NC program or press CANCEL or START or only START to continue the NC program without any changes.

Program continuation by: Press the "Cancel" key to cancel the alarm. No further operation required.

10900

Channel %1 block %2 No S value programmed for constant cutting rate

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

If G96 is active, the constant cutting rate under address S is missing.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.

Remedy: Either program a constant cutting rate under S in [m/min] or deselect function G96. For example, with G97, the previous feedrate remains stored, but the spindle will go on rotating at the current speed.

Program continuation by: Press NC START to cancel the alarm and continue the program.

10910 Channel %1 block %2 Extreme velocity increase in one path axis

Explanation
%1 = channel number
%2 = block number, label

With transformation selected, an extreme increase in the velocity occurs in one or several axes, e.g. since the path runs in the vicinity of the pole.

Reaction
Alarm display.

Remedy
Subdivision of the NC block into several blocks (e.g. 3) to keep the path section with the increased speed as short as possible, achieving a short time for it. In this case, the remaining blocks will be traversed at the programmed velocity.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

10911 Channel %1 block %2 Transformation does not permit passing the pole

Explanation
%1 = channel number
%2 = block number, label

The specified course of the curve passes through the pole of the transformation.

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy
Modify the part program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10914 Motion not possible with transformation active - in channel %1, block %2

Explanation
%1 = channel number
%2 = block number, label

The specified motion is not possible due to the machine kinematics. Transformation-depending error causes may be:
TRANSMIT:
A (circular) area exists around the pole to which positioning is not possible. This area is due to the fact that the tool reference point cannot be traversed into the pole.

The area is defined by:
- the machine data ($MC_TRANSMIT_BASE_TOOL..)
- the active tool length compensation (see $TC_DP..).
  - How the tool length compensation is taken into account depends on the work plane selected (see G17,..).
- The machine will stop before the faulty block.

Reaction
Alarm display
Interface signals are set
NC Start inhibited

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

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10930

Channel %1 block %2 Illegal type in stock removal contour

Explanation

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

The contour of the stock removal cycle (CYCLE 95) contains travel commands other than: G00, G01, G02, G03, CIP or CT. The contour program may only contain contour elements consisting of these G commands (i.e., no thread blocks, no spline blocks, etc.).

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

Program only path elements for the stock removal contour, which consist of straight lines and arcs.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10931

Channel %1 block %2 Error in programmed stock removal contour

Explanation

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

The stock removal contour (CYCLE 95) contains the following errors:

- full circle
- intersecting contour elements
- wrong start position

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

Correct the errors mentioned above in the program for the contour to be machined.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10932

Channel %1 block %2 Preparation of contour has been restarted

Explanation

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

The stock removal cycle CYCLE 95 has been interrupted during the preparation phase of the stock removal contour.

Reaction

Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy

Do not admit interruption in the stock removal cycle CYCLE 95 during the contour preparation.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

10933

Channel %1 block %2 Contour program contains too few contour blocks

Explanation

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

The program in which the stock removal contour is programmed contains less than 3 blocks with motions in both axes in the machining plane. The stock removal cycle (CYCLE 95) was canceled.

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy
Enlarge the program with the stock removal contour to at least 3 NC blocks with axis motions in both axes of the current machining plane.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

10934 Channel %1 block %2 Array for contour segmentation is dimensioned too small

Explanation
%1 = channel number
%2 = block number, label

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

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy
Reduce the number of blocks in the contour program. Check the division of the contour into several programs.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

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

Explanation
%1 = channel number
%2 = block number, label
%3 = source string of address

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

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

- Remove addresses from the NC program which are used repeatedly (except those for which repeated value assignments are allowed).
- Check whether the address (e.g. the axis name) is specified via a user-defined variable (possibly not easily to be seen if the assignment of the axis name to the variable is only carried out by arithmetic operations in the program).

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12010 Channel %1 block %2 address %3 Address type programmed too often

Explanation
%1 = channel number
%2 = block number, label
%3 = source string of address

It is defined for each address type how often it may occur in an NC block (for example, all axes together form an address type, which is also subject to a block limit).

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Split the program information over several blocks. (When doing so, however, pay attention to non-modal functions.).

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12020

Channel %1 block %2 Illegal address modification

Explanation
%1 = channel number
%2 = block number, label

Valid address types are ‚IC', ‚AC', ‚DC', ‚ACN', ‚ACP'. Not every of these address modifications can be applied to every address type. Which of them can be used for the individual address types, is specified in the Programming Guide. If this address modification is applied to illegal address types, this alarm is generated, e.g.:

N10 G02 X50 Y60 **I=DC(20)** J30 F100
; Interpolation parameter with DC.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Apply non-modal address modifications only to permissible addresses, as specified in the Programming Guide.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12050

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

Explanation
%1 = channel number
%2 = block number, label
%3 = NC address in source text block

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

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Study the Programming Guide and the machine data with reference to the addresses really configured and their meaning and correct the NC block accordingly.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12060

Channel %1 block %2 Same G group programmed repeatedly

Explanation

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

The G functions that can be used in the part program are divided into groups which are **syntax-defining** or **non syntax-defining**. Only **one G function** each from each G group may be programmed. The functions within a group are mutually exclusive.

The alarm concerns only the non-syntax defining G functions. If several G functions from these groups are called in an NC block, the **last** function of a group will be active (all previous functions are ignored).

G FUNCTIONS:

Syntax-defining G functions	Non-syntax defining G functions
1st to 4th G groups	**5th to nth G groups**

Assignment of the G groups

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

No remedial action required. However, it should be checked whether the last programmed G function is really the desired one.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12070

Channel %1 block %2 Too many syntax-defining G functions

Explanation

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

Syntax-defining G functions define the structure of the part program block and of the addresses contained in it. **Only one** syntax-defining G function may be programmed in an NC block. The G functions of the 1st ... 4th G groups are syntax-defining.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Analyze the NC block and distribute the G functions over several NC blocks.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12080

Channel %1 block %2 Syntax error in text %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source text area

The grammar of the block has been violated at the displayed text position. The exact error cause cannot be specified more exactly, since there are too many possible error causes.

Example 1:
N10 IF GOTOF ... ; the jump condition is missing!

Example 2:
N10 R-50 =12 ; faulty arithmetic parameter number

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Analyze the block and correct it using the syntax diagram specified in the Programming Guide.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12090

Channel %1 block %2 Parameter %3 not expected.

Explanation
%1 = channel number
%2 = block number, label
%3 = illegal parameter in the text

The programmed function is predefined and does not allow parameters in its call. The first unexpected parameter will be displayed.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Program a function without parameter transfer.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12100

Channel %1 block %2 Number of passes %3 not permitted

Explanation
%1 = channel number
%2 = block number, label
%3 = number of passes

Subroutines called using **MCALL** are modally effective, i.e. each block containing positional data is automatically followed by a **single** subroutine pass. The programming of a number of passes under address P is therefore not admissible.

The modal call will act until a new MCALL is programmed, either with or without a new subroutine name (deletion function).

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Program the subroutine call without number of passes.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12110 Channel %1 block %2 Block syntax cannot be interpreted

Explanation

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

The addresses programmed in the block are not allowed with the valid syntax-defining G function.

Example:G1 I10 X20 Y30 F1000;
It is not permitted to program linear block parameters in the linear block.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Check the block structure and correct it as required by the program.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12120 Channel %1 block %2 G function not programmed separately

Explanation

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

The G function programmed in this block must be written in a separate block. General addresses or synchronous actions may not occur in the same block. These G functions are:

G25, G26	Working area/spindle speed limitation
G110, G111, G112	Pole programming with polar coordinates

Example:G4 F1000 M100: It is not permitted to program M functions in a G4 block.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Program the G function in a separate block.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

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

Explanation

%1 = channel number
%2 = block number, label
%3 = software construct in source text

The full version of the control system provides functions which are not implemented in the current variant.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Remove the displayed function from the program.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12150 Channel %1 block %2 Operation %3 not compatible with this data type

Explanation

%1 = channel number
%2 = block number, label
%3 = string (violating operator)

The data types are not compatible with the required operation (within an arithmetic expression or a value assignment).

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Modify the definition of the variables used such that the desired operations can be carried out.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12160 Channel %1 block %2 Range of values exceeded

Explanation

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

The constant programmed for the variable exceeds the value range which has been defined by the data type in advance.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Correct the value of the constants or adapt the data type accordingly. If the value for an integer constant is too large, it can also be specified by appending a decimal point as a real constant.

Example:
R1 = 9 876 543 210 correct to: R1 = 9 876 543 210.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12170 Channel %1 block %2 Name %3 defined repeatedly

Explanation

%1 = channel number
%2 = block number, label
%3 = symbol in the block

The symbol used in the error message has already been defined in the running part program.

Please take into account that user-defined identifiers may occur repeatedly if the multiple definition is carried out in other programs (subroutines), i.e. local variables with the same name may be defined again if the program (subroutine) has been quitted or is already completed.
This applies both to user-defined symbols (labels, variables) and for machine data (axes, DIN addresses and G functions).

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy
The symbol that the database management already knows will be displayed. Use the program editor to search for this symbol in the definition part of the current program. The 1st or 2nd symbol must be assigned different names.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12180 Channel %1 block %2 Illegal linking of operators %3

Explanation
%1 = channel number
%2 = block number, label
%3 = linked operators

The term 'operator linking' denotes binary and unary operators written one after the other, without using brackets.

Example:
N10 R1=R2-(-R3) ; Correct notation
N10 R1=R2--R3 ;error!

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Formulate the expression correctly and unambiguously using brackets; this will increase both the clarity and the legibility of a program.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12200 Channel %1 block %2 symbol %3 cannot be created

Explanation
%1 = channel number
%2 = block number, label
%3 = symbol in the source block

The symbol to be created using the DEF statement cannot be created, as:

- it is already defined (e.g. as a variable or function)
- the internal memory is no longer sufficient (e.g. in case of large fields)

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Carry out the following checks:

- Use the text editor to check whether the name to be assigned has already been used in the current program cycle (main program and called subroutines).
- Estimate how much memory is required by the symbols already defined; one possibility of reducing it is to use fewer global and more local variables.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12260 Channel %1 block %2 Too many initialization values specified%3

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

When initializing a field (field definition and value assignment to individual field elements), more initialization values exist than field elements.

Example:
N10 DEF INT OTTO[2,3]=(..., ..., {more than 6 values})

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Check in the NC program whether:

1. the number of field elements (n,m) has been specified correctly when defining the field elements (DEF INT FELDNAME[n,m], e.g. a field with 2 lines and 3 columns: n=2, m=3).
2. the value assignment has been carried out correctly when initializing (the values of the individual field elements are separated by a **comma**, variables of the type REAL by a **decimal point**)

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12261

Channel %1 block %2 %3 may not be initialized

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

Variables of the type "frame" may not be initialized in the definition - example:
DEF FRAME LOCFRAME = CTRANS(X,200).

It is also not possible to define default values when initializing fields in the program sequence with axes using SET.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
carry out the initialization in a separate block in the execution part of the program:
DEF FRAME LOCFRAME LOCFRAME = CTRANS(X,200)

When using axis variables:
DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = SET (X, , Y) - to be replaced by:
DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = X AXIS_VAR [7] = Y

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12290

Channel %1 block %2 Arithmetic variable %3 not defined

Explanation
%1 = channel number
%2 = block number, label
%3 = source string of arithmetic variable

Only the R parameters are predefined as arithmetic variables - all the other arithmetic variables must be defined before using them with the DEF instruction. The number of arithmetic parameters is defined via machine data. The names must be unambiguous and only be used in the control system once (exception: local variable).

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Define the desired variable in the definition part of the program (possibly in the calling program if you wish to program a global variable).

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12300

Channel %1 block %2 Call-by-reference parameter missing in subroutine call %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

In the subroutine definition, a formal **REF parameter (call-by-reference parameter)** has been specified, which no current parameter has been assigned.

The assignment is carried out when calling the subroutine because of the position of the variable name and not because of the name!

Example:
Subroutine: (2 call-by-value parameters X and Y,
1 call-by-reference parameter Z)

```
PROC XYZ (INT X, INT Y, VAR INT Z)
:
M17
ENDPROC
```

Main program:

```
N10 DEF INT X
N11 DEF INT Y
N11 DEF INT Z
:
N50 XYZ (X, Y)   ;REF parameter Z missing
or
N50 XYZ (X, Z)   ;REF parameter Y missing
```

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Assign all REF parameters (call-by-reference parameters) of the subroutine a variable when calling. The standard formal parameters (call-by-value parameters) need not be assigned a variable; these are assigned 0 by default.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12320

Channel %1 block %2 parameter %3 is no variable

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

When calling the subroutine, a REF parameter has been assigned not a constant, but a constant or the result of a mathematic expression even if only variable identifiers are permitted.

Examples:
N10 XYZ (NAME_1, **10,** OTTO) or
N10 XYZ (NAME_1, **5 + ANNA,** OTTO)

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Remove the constant or the mathematic expression from the NC block.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12330

Channel %1 block %2 Invalid parameter type %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

When calling a procedure (of a subroutine), it is detected that the type of the actual parameter cannot be converted into the type of the formal parameter. There are 2 possible cases:

- **Call-by-reference parameter:** Actual parameter and formal parameter must be exactly of the same type, e.g. STRING, STRING.
- **Call-by-value parameter:** Generally, actual parameter and formal parameter could be different if a conversion would basically be possible. In the present case, however, the types are generally not compatible, e.g. STRING → REAL.

to from	REAL	INT	BOOL	CHAR	STRING	AXIS	FRAME
REAL	yes	yes*	yes[1)]	yes*	-	-	-
INT	yes	yes	yes[1)]	if value 0...255	-	-	-
BOOL	yes	yes	yes	yes	-	-	-
CHAR	yes	yes	yes[1)]	yes	yes	-	-
STRING	-	-	yes[2)]	only if 1 character	yes	-	-
AXIS	-	-	-	-	-	yes	-
FRAME	-	-	-	-	-	-	yes

Table Type conversion

[1)] Value <>0 corresponds to TRUE, value ==0 corresponds to FALSE.
[2)] String length 0 => FALSE, otherwise TRUE
[*)] When converting the type from REAL to INT, it is rounded up with a fraction value >=0.5, otherwise, it is rounded off

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Check the transfer parameter of the subroutine call and define it as a call-by-value or call-by-reference parameter, depending on its use.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12340

Channel %1 block %2 Too many parameters %3

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

When calling a function or a procedure (whether predefined or user-defined), more parameters have been transferred than defined.

Predefined functions and procedures:
The number of parameters is fixed in the NC.

User-defined functions and procedures:
The number of parameters (via type and name) is defined when defining the parameters.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey.The correction cursor will be positioned on the block to be corrected.

Check whether the right procedure/function has been called. Program the number of parameters according to the procedure/function.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12360

Channel %1 block %2 Invalid parameter dimension %3

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

Check the following error possibilities:

1. The current parameter is a field, but a formal parameter is a variable.
2. The current parameter is a variable, but a formal parameter is a field.
3. The current and the formal parameters are fields, but with dimensions not to be agreed.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Correct the NC part program depending on the error cause specified above.

Press NC START to cancel the alarm and continue the program execution.

Program continuation by

12380 Channel %1 block %2 Maximum memory capacity reached

Explanation

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

The data definitions of this block cannot be carried out, since the maximum available memory provided by the data management is exhausted or cannot store any more data.
This alarm may also occur if several subroutine calls are processed in succession, without generating a block with effect on the machine (motion, dwell time, M function).

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Please inform the authorized personnel/customer service.
Reduce the number of variables, reduce the fields or enlarge the capacity of the database management.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12400 Channel %1 block %2 field %3 Element does not exist

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

This can have the following causes:
- Illegal index list; an axis index is missing
- Field index does not match with variable definition
- When initializing the field, you tried to access a variable not using the standard access with
  SET or REP.
  Single character access or missing indices are not possible.

When initializing this field, you addressed a field that does not exist.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

- **Field initialization:** Check the field index of the addressed element. The 1st field element will receive index [0,0], the 2nd [0,1], etc. The right field index (column index) will be incremented first.

This means that the 2nd row, the 4th element is addressed via index [1,3] (The indices start with zero).

- **Field definition:** Check the field size. The 1st number specifies the number of elements in the 1st dimension (number of rows), the 2nd number the number of elements in the 2nd dimension (number of columns).

A field with 2 rows and 3 columns must be defined using the specification [2,3].

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12410 Channel %1 block %2 Invalid index type in text %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

When assigning a value to an element of a field variable, the field index was specified illegally.

Only the following is permitted as a field index (in square brackets):

- **Axis identifier** if the field variable was defined as data type FRAME
- **Integer values** with all the other data types.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Correct the indices of the field element with reference to the variable definition or define the field variable otherwise.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12420 Channel %1 block %2 Identifier %3 too long

Explanation

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

The symbol to be defined or the specified jump destination possesses a name which is longer than the 32 characters permitted.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

The symbol to be created or the jump destination with program jumps (label) must be selected within the system conventions, i.e. the name must start with two letters (but the 1st character may not be a § character) and may not be longer than 32 characters.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12430 Channel %1 block %2 Invalid index specified

Explanation

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

When specifying the array index (in the field definition), an index was used which is outside the permissible range.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Specify the field index within the permissible range.
Range of values per field dimension: 1 ... 32 767.

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

12440

Channel %1 block %2 Maximum number of formal parameters exceeded

Explanation:

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

When defining a procedure (of a subroutine) or an EXTERNAL statement, more than 127 formal parameters were specified.

Example:PROC ABC (FORMPARA1, FORMPARA2, ...

... FORMPARA127, **FORMPARA128, ...**)

EXTERN ABC (FORMPARA1, FORMPARA2, ...

... FORMPARA127, **FORMPARA128, ...**)

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

Remedy: Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Check whether really all parameters must be transferred. If so, a reduction of the number of formal parameters can be achieved by using global variables or R parameters or by summarizing parameters of the same type in an array and transferring them in this way.

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

12450

Channel %1 block %2 Label defined repeatedly

Explanation:

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

The label of this block already exists.

If you compile the NC program off-line, the entire program is compiled block by block. Multiple designations are detected with 100 % safety, what in the case of **online compilation** is not necessarily the case. (Only the current program sequence is compiled here, i.e. program branches currently not passed will not be considered and may therefore have programming errors).

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

Remedy: Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction pointer is positioned on the block in which the displayed label occurs for the second time.

Use the editor to browse the part program where the designation you are looking for occurs for the 1st time, and change one of the two names.

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

12460 Channel %1 block %2 Maximum number of symbols containing %3 exceeded

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

The max. number of variable definitions, cycle programs, cycle parameters, which can be stored in the data management of the control system, has been exceeded.
If the alarm occurs in conjunction with 15175 (cycles have been reloaded), the memory provided is not sufficient. A possible remedy is to modify the machine data.
If this alarm occurs in conjunction with alarm 15180 (downloading of initial.ini failed), the name of the block that caused the error is to be found in this alarm. (for a list of the names and their meaning -> see Documentation on alarm 6010)

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Please inform the authorized personnel/customer service.
Reduce the number of symbols in the block (possibly also by making use of the array technique or by using R parameters) or adapt the machine data accordingly (if you possess the appropriate access right).

GUD blocks may cause errors only when downloading the initial.ini file.

Cycle program definitions are reloaded with each POWER ON/NC RESET. That means these blocks may cause errors only in conjunction with this process.
See also the explanations on alarm 6010.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12470 Channel %1 block %2 Unknown G function %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

A non-defined G function has been programmed in the displayed block.

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Decide on the basis of the Programming Guide of the machine manufacturer whether the G function displayed is generally not existing or a reconfiguration of a standard G function has been carried out.

Remove the G function from the part program or program the function call as specified in the Programming Guide of the machine manufacturer.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12475 Channel %1 block %2 Invalid G function number %3 programmed

Explanation

%1 = channel number
%2 = block number, label.
%3 = G code number

With indirect G code programming, an illegal G function number (parameter 3) was programmed for a G group.
Only the G function numbers specified in the Programming Guide, Section 12.3 "List of G functions/preparatory functions" are permitted.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy: Correct the part program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

12480 Channel %1 block %2 Subroutine %3 already defined

Explanation:

%1 = channel number
%2 = block number, label
%3 = source string

The name used in the PROC or EXTERN statement is already defined in another call description (e.g. for cycles).

Example:
EXTERN **CYCLE85** (VAR TYP1, VAR TYP2, ...)

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

Remedy: Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Select a program name that has not yet been used as an identifier. (Theoretically, it would also be possible to match the parameter declaration of the EXTERN statement to the existing subroutine to avoid the alarm. In this case, however, the definition would be carried out twice completely identically).

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

12490 Channel %1 block %2 Illegal access right %3

Explanation:

%1 = channel number
%2 = block number, label
%3 = source string

The desired access level has not been set. The desired protection class is out of the admissible value range.

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

Remedy: Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

- Using the operator panel, set the current protection level at least to the level at which the variable will have the highest level.
- Program the protection level within permitted range of values.
- Program new protection levels below the old values.

Program continuation by: Press NC START to cancel the alarm and continue the program execution.

12500 Channel %1 block %2 %3 is not possible in this block

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

The displayed vocabulary word may not be used in this type of block and at this position (the term 'blocks' covers all blocks that occur in the NC).

Types of blocks:

Program block

contains a main program and a subroutine

Data block

contains macro and variable definitions, as well as calls an M, H or E function

Initialization block

contains only selected language elements for data initialization

Reaction

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Remove the displayed language element (vocabulary word) including its parameters from this block and insert it in the relevant block.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12520 Channel %1 block %2 Too many tool data %3

Explanation

%1 = channel number
%2 = block number, label
%3 = source string

A maximum of 5 tool compensation parameters may be used per block in the part program, tool compensation file (..._TOA) and in the initialization file (..._INI).

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

Alarm display.
Interface signals are set.
Correction block.

Remedy

Press NC STOP and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

- Split the part program block over several blocks.
- If necessary use local variables for storing intermediate results.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

12540 Channel %1 block %2 is too long or too complex

Explanation
%1 = channel number
%2 = block number, label

The maximum internal block length after processing by the translator may not exceed 200 characters.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Split the program block into several partial blocks.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12550 Channel %1 block %2 Name %3 not defined or option not existing

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

The displayed identifier has not yet been defined prior to its use.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

- Correct the used name (typing mistake)
- Check the definition of variables and subroutines
- Check the options.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12560 Channel %1 block %2 Programmed value %3 out of permissible limits

Explanation
%1 = channel number
%2 = block number, label
%3 = source string

When assigning the values, the admissible range of values for the data type has been exceeded.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Press the NC STOP key and select the "Correction block" function using the PROGRAM CORRECTION softkey. The correction cursor will be positioned on the block to be corrected.

Carry out the value assignment within the range of values permitted for the particular data type; if necessary use a different type to enlarge the range of values.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12590 Channel %1 block %2 Cannot create global user data

Explanation
%1 = channel number
%2 = block number, label

The number of global user data blocks is defined in the machine data.
The directory _N_DEF_DIR contains a file with definitions for global user data whose block number is greater than the number of blocks specified in the MD.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Please inform the authorized personnel/customer service.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12600 Channel %1 block %2 Invalid line checksum

Explanation
%1 = channel number
%2 = block number

When executing an INI file or executing a TEA file, an invalid line checksum has been detected.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the INI file or correct the MD and create a new INI file (via 'upload').

Program continuation by
Power ON

12630 Channel %1 block %2 Illegal skipping label / label in control structure

Explanation
%1 = channel number
%2 = block number

Blocks that contain control structures (FOR, ENDIF, etc.) may not be skipped and may not contain labels.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Correct the part program.
Simulate a skipping label by IF request. Write the label in a separate block before the control structure block.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

12640 Channel %1 block %2 Nesting conflict with control structures

Explanation
%1 = channel number
%2 = block number

Error in program sequence: Opened control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) are not completed, or there is no loop start to the programmed loop end.

Example:
LOOP ENDIF ENDLOOP

Reaction
Alarm display.
Interface signals are set.

	Interpreter stop. NC Start inhibited.
Remedy	Correct the part program such that also all opened control structures are completed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

12641 Channel %1 block %2 Max. nesting depth for control structures exceeded

Explanation	%1 = channel number %2 = block number Max. nesting depth of control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) exceeded. The max. nesting depth is currently 8.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Correct the part program. If necessary relocate parts to a subroutine.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

12661 Channel %1 block %2 technology cycle %3:No further subroutine call possible

Explanation	%1 = channel number %2 = block number %3 = name of technology cycle call It is not possible to call a subroutine or another technology cycle in a technology cycle.
Reaction	Alarm display. Interface signals are set. Correction block.
Remedy	Modify the part program.
Program continuation by	Use the RESET key to cancel the alarm.

12700 Channel %1 %2 Programming of a contour definition not allowed, since modal subroutine active

Explanation	%1 = channel number %2 = block number, label A block containing a contour definition is programmed in the external language mode, and at the same time, a modal cycle is active. Contour definition programming may not be used in external language mode if a modal cycle is active because of ambiguous address assignment (e.g. R=radius for contour definition or retraction plane for drilling cycle).
Reaction	Alarm display. Interface signals are set. Correction block
Remedy	Modify the part program.
Program continuation by	Use the "Cancel" key to cancel the alarm.

12701 Channel %1 block %2 Illegal interpolation type for contour definition active

Explanation
%1 = channel number
%2 = block number, label

In a contour definition block, not G01 is active as the interpolation function. In a contour definition block, the linear interpolation must always be selected with G01. G00, G02, G03, G33 etc. are not permitted.

Reaction
Alarm display.
Interface signals are set.
Correction block

Remedy
Modify the part program.Program linear interpolation G01.

Program continuation by
Press NC START to cancel the alarm and continue the program.

12710 Channel %1 block %2 Illegal language element in external language mode

Explanation
%1 = channel number
%2 = block number, label

The programmed language element is either not permitted in the external language mode or is unknown. In the external language mode, only SIEMENS mode language elements are permitted, which are used for subroutine calls (except Lxx), and the language constructs for repeating parts of the program using REPEAT (UNTIL):

Reaction
Alarm display.
Interface signals are set.
Correction block

Remedy
Modify the part program.

Program continuation by
Use the "Cancel" key to cancel the alarm.

12722 Channel %1 block %2 Several ISO_2/3 macro or cycle calls in the block

Explanation
%1 = channel number
%2 = block number, label

Cycle and macro calls are programmed in a block together, e. cycle calls using G81 ... G89 together with an M macro in a block or G65/G66 macro calls together with M macros in a block.
Only one macro or cycle call is admitted in an NC block.

Reaction
Alarm display.
Interface signals are set.
Correction block

Remedy
Distribute cycle and macro calls over several blocks.

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

14000 Channel %1 block %2 Illegal end of file

Explanation
%1 = channel number
%2 = block number, label

M02 or **M30** is expected as the end of files of main programs; in subroutines **M17.** The block preparation (data management) does not provide a subsequent block even if no end of file was programmed in the previous block.

Reaction
Alarm display.
Interface signals are set.

	Interpreter stop. NC Start inhibited.
Remedy	Check whether the end of program has been forgotten to be entered, or whether a jump to a program section which contains the end-of-block character is carried out in the last program block.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14001 Channel %1 block %2 Illegal end of block

Explanation	%1 = channel number %2 = block number, label A system-internal data manipulation (e.g. reloading from an external) can be followed by a partial file, without LF as the last character.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Read out the part program and use a text editor to modify the program (e.g. insert blanks or comments in front of the displayed block) to have a modified structure of the part program when re-reading the part program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14010 Channel %1 block %2 Illegal default parameter in subroutine call

Explanation	%1 = channel number %2 = block number, label When calling a subroutine with parameter transfer, parameters have been omitted which cannot be replaced by default parameters (call-by-reference parameters or parameters of the type AXIS. The remaining missing parameters is assigned the value 0 or frames from the unit frame by default).
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Assign the parameters not yet defined values in the subroutine call.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14011 Channel %1 block %2 Program %3 does not exist or not released for execution

Explanation	%1 = channel number %2 = block number, label %3 = program name The called program (main program or subroutine) has been called from the part program currently running (main program or subroutine). However, either it does not exist in the NC memory, or the option for the used function is not enabled.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Correct the NC part program.

1. Check the subroutine name in the calling program.
2. Check the name of the called program
3. Check whether the program has been transferred into the NC memory.
4. Check the options or upgrade/set them.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14012 Channel %1 block %2 Maximum subroutine level exceeded

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

The maximum nesting depth of 8 program levels has been exceeded.

Subroutines that have a maximum nesting depth of 7 subroutines can be called from the main program.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Modify the machining program such that the nesting depth is reduced, e.g. use the editor to copy a subroutine of the next nesting depth into the calling program and remove the call for this subroutine. This will reduce the nesting depth by one program level.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14013 Channel %1 block %2 Illegal number of subroutine passes

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

When a subroutine is called, the programmed number of passes P is either zero or negative.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Program a number of passes between 1 and 9 999.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14014 Channel %1 Selected program or access rights do not exist

Explanation: %1=channel number

The selected part program is not in the NC memory.

Reaction: Alarm display.

Remedy: Reload the desired program into the NC memory or check the name of the directory (workpiece overview) and correct it.

Program continuation by: Use the "Cancel" key to cancel the alarm. Restart the part program.

14015 Channel %1:No access rights for this file

Explanation: %1=channel number

The user has no execution rights for the file.

Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Alter the user rights.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14016 Channel %1 block %2 Error calling subroutine using M/T function

Explanation	%1 = channel number %2 = block number, label The following conflict has been found when calling a subroutine using an M or T function: In the block referenced using the parameter %2, - a replacement of an M or T function has already been activated - a modal subroutine call is active - a subroutine return is programmed - an M98 subroutine call is active (only on external language mode)
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Principally, the M or T function can only be replaced if a subroutine call or subroutine return is not already carried out due to other program constructs. The part program must be corrected accordingly.
Program continuation by	Use the "Cancel" key to cancel the alarm.

14017 Channel %1 block %2 Syntax error calling subroutine using M/T function

Explanation	%1 = channel number %2 = block number, label When calling the subroutine using an M function with parameter transfer, an illegal syntax has been detected: - Address extension not programmed as a constant. - M function value not programmed as a constant. Note: If a parameter transfer has been programmed for an M function replacement via MD $MN_M_NO_FCT_CYCLE_PAR, the limitation applies to this M function that both an address extension and an M function value must be programmed as a constant when replacing.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the programming of the M function.
Program continuation by	Press NC START to cancel the alarm and continue the program.

14020

Channel %1 block %2 Invalid value or illegal number of parameters when calling the function or procedure

Explanation

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

- When calling a function or procedure, an illegal parameter value has been specified.
- When calling a function or procedure, an illegal number of actual parameters has been programmed.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy

Modify the part program.

Program continuation by

Press NC START to clear the alarm. Restart the part program.

14021

Channel %1 block %2 Invalid value or illegal number of parameters when calling the function or procedure

Explanation

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

- When calling a function or procedure, an illegal parameter value has been specified.
- When calling a function or procedure, an illegal number of actual parameters has been programmed.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Modify the part program.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14040

Channel %1 block %2 Error in end point of circle

Explanation

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

With circular interpolation, the distance either between the circle radii for start point and end point or the distance between the circle center points is greater than defined in the machine data.

1. With radius programming, start and end points are identical so that the position of the circle is not defined by start or end point.
2. **Radii:** The NC will calculate the radii for start and end points from the current start point and the remaining programmed circle parameters. The alarm message is provided if the difference of the circle radii is either
   - greater than the value in MD 21000 **CIRCLE_ERROR_CONST** (in case of small radii if the programmed radius is less than the quotient of MD 21000 **CIRCLE_ERROR_CONST** divided by MD 21010 **CIRCLE_ERROR_FACTOR)**, or
   - greater than the value in MD 21010 **CIRCLE_ERROR_CONST** (in case of large radii if the programmed radius is greater than the quotient of MD 21000 **CIRCLE_ERROR_CONST** divided by MD 21010 **CIRCLE_ERROR_FACTOR**).

3. **Center points:** A new circle center will be calculated using the circle radius to the starting point. The circle center will be located on the mid-perpendicular established on the connecting straight line of circle start and circle end points. The angle, which is specified in arc dimension, between the two straight lines from the starting point to the center point either calculated in the way described or programmed must be less than the root from 0.001 (corresponds to approx. 1.8 degrees).

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Please inform the authorized personnel/customer service.
Check MD 21000 **CIRCLE_ERROR_CONST** and MD 21010 **CIRCLE_ERROR_FACTOR**. If the values are within reasonable limits, program the end-of-circle point or the circle center point more exactly.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14045

Channel %1 block %2 Error programming a tangential circle

Explanation
%1 = channel number
%2 = block number, label

This alarm may have the following causes:

- When programming a tangential circle, you did not have programmed the tangent direction, e.g. since prior to the current block no other traversing block was programmed.
- A circle can be generated from start and end points, as well as from the tangent direction, since viewed from the start point, the end point is in the opposite direction specified by the tangent.
- No tangential circle can be generated, since the tangent stands vertically on the active plane.
- In the special case that the tangential circle changes to a straight line, several full circle turns have been programmed using TURN.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Start inhibited.
NC Stop at alarm at block end.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14048

Channel %1 block %2 Invalid number of revolutions in circle programming

Explanation
%1 = channel number
%2 = block number, label

When programming a circle, a negative number of full revolutions has been specified.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop
NC Start inhibited

Remedy
Modify the part program.

Program continuation by

14050 Channel %1 block %2 Nesting depth for arithmetic operations exceeded

Explanation
%1 = channel number
%2 = block number, label

To calculate arithmetic expressions in NC block, an operand stack of fixed size is used. With very complex expressions, this stack might overflow.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Distribute complex arithmetic expressions over several, more simply structured arithmetic blocks.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14051 Channel %1 block %2 Arithmetic error in part program

Explanation
%1 = channel number
%2 = block number, label

- With the calculation of an arithmetic expression, an overflow occurred (e.g. division by zero).
- The value range that can be represented for a certain data type has been exceeded.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy
Analyze the program and correct the faulty program section.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14060 Channel %1 block %2 Illegal skip level with split block skipping

Explanation
%1 = channel number
%2 = block number, label

With "skipped block skipping", a skip level greater than 1 has been specified (In package 1, the specification of a value for the skip level is already denied by the converter as a **syntax error**, i.e. only one Skip Block level ON/OFF is possible).

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Enter skip level (number after slash) 1.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14070 Channel %1 block %2 Insufficient variable memory for subroutine call

Explanation

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

A subroutine you have called cannot be executed (opened) since either the internal data memory generally to be created is not sufficient or the available local memory space is too small for the local program variables.
This alarm may only occur in MDA.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Analyze the relevant section of the part program:

1. Was always the optimum data type selected for variable definitions? (e.g. REAL for data bits would be bad - better: BOOL)
2. Could local variables be replaced by global variables?

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14080 Channel %1 block %2 Jump destination not found

Explanation

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

The jump target of conditioned and unconditioned jumps with in the program must be a block with a **label** (symbolic name instead of block number). If no jump target with the specified label is found when searching **in the programmed direction**, this alarm is displayed.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Check the NC program for the following possible errors:

1. Check whether the target designation is identical to the label.
2. Is the jump direction correct?
3. Has the label been completed with a colon?

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14082 Channel %1 block %2 Program section not found

Explanation

%1 = channel number
%2 = block number, label
%3 = start or end label

Either the starting point for the program repetition with CALL <program name> BLOCK <start label> TO <end label> was not found or the same program part repetition was called recursively.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy

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

Program continuation by

Press NC START or RESET to cancel the alarm and continue the program.

14085 Channel %1 block %2 Illegal statement

Explanation
%1 = channel number
%2 = block number, label

The statement 'TML()' may only used in the subroutine using the T command.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm at end of block.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14088 Channel %1 block %2 axis %3 Invalid position

Explanation
%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

An axis position was programmed which is greater than 3.40e+38 increments.

Reaction
Alarm display.
Interface signals are set.
Correction block.

Remedy
Modify the part program.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14091 Channel %1 block %2 Illegal function, index:%3

Explanation
%1 = channel number
%2 = block number, label
%3 = index

A function was programmed or initiated which is not permitted in the context of the current program. The function in question is encrypted in the parameter "Index":
Index = 1: "RET" command was programmed on the main program level.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Index = 1: Replace the RET command by M17/M30.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14092 Channel %1 block %2 axis %3 Illegal axis type

Explanation
%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the part program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14095

Channel %1 block %2 Radius too small for radius programming

Explanation

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

On radius programming, a radius too small has been specified, i.e. the programmed radius is less than the half distance between start and end points.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy

Modify the part program.

Program continuation by

Press NC START to cancel the alarm and continue the program.

14096

Channel %1 block %2 Type conversion not permitted

Explanation

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

During the program execution, data have been linked due to a variable/value assignment or due to an arithmetic operation such that they have to be converted to another type. When doing so, the range of values would be exceeded.

Variable Type	Property	Range of Values
REAL	Fractional numbers with decimal point	**$\pm (2^{-1022}-2^{+1023})$**
INT	Integer numbers with sign	**$\pm (2^{31}-1)$**
BOOL	Truth value TRUE, FALSE	**0,1**
CHAR	1 ASCII character	**0 - 255**
STRING	String (max. 100 values)	**0 - 255**
AXIS	Axis addresses	**Only axis names**
FRAME	Geometrical specifications	**as distances traversed by the axes**

Ranges of values for the individual variable types

to / **from**	**REAL**	**INT**	**BOOL**	**CHAR**	**STRING**
REAL		yes [1]	yes	yes [2]	-
INT	yes		yes	yes [2]	-
BOOL	yes	yes		yes	-
CHAR	yes	yes	yes		yes
STRING	-	-	yes	yes [3]	

Type conversion

[1] Value <>0 corresponds to TRUE, value ==0 corresponds to FALSE.
[2] String length 0 => FALSE, otherwise TRUE
[3] if only 1 character

No conversion can be carried out from and to the types AXIS and FRAME.

Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the program part such that the range of values is not exceeded, e.g. change the variable definition.
Program continuation by	Press NC START to cancel the alarm and continue the program.

14098 Channel %1 block %2 Conversion error: No valid number found

Explanation	%1 = channel number %2 = block number, label The string is not a valid INT or REAL number.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Modify the part program. If it is an input, it is possible to check via the predefined function ISNUMBER (with the same parameter) whether the string constitutes a number.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14130 Channel %1 block %2 Too many initialization values specified

Explanation	%1 = channel number %2 = block number, label When assigning the field via SET, more initialization values have been specified for the program execution than field elements exist.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Reduce the number of initialization values.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14160 Channel %1 block %2 No geometry axis specified for tool length compensation

Explanation	%1 = channel number %2 = block number, label If with tool length compensation using H word and G43/G44 in ISO_mode variant C is activated via MD 20380 **TOOL_CORR_MODE** (the tool length is effective along the programmed axis), exactly one geometry axis must be programmed together with H if MD 20384 **TOOL_CORR_MULTIPLE_AXES** = TRUE is not set. This alarm is output if either none or more than one geometry axis is programmed together with H. It is permitted to program several axes if the MD 20384 **TOOL_MULTIPLE_AXES** = TRUE was set. If no axis is specified, this is always an error.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block. NC Stop at alarm.
Remedy	Change MD 20380 **MCTOOL_CORR_MODE** or modify the part program.

Program continuation by

14165 Channel %1 block %2 Tool compensation selected without active tool

Explanation
%1 = channel number
%2 = block number, label

If a tool compensation is activated in the ISO M language mode(G43/G44), a tool number (H) must be specified.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14170 Channel %1 block %2 Illegal interpolation type with tool length compensation

Explanation
%1 = channel number
%2 = block number, label

If a tool compensation is activated in the ISO M language mode(G43/G44), linear interpolation type must be activated.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14180 Channel %1 block %2 H number not defined

Explanation
%1 = channel number
%2 = block number, label

The specified H number is not assigned a tool (ISO MO).

Reaction
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.
Alarm display.

Remedy
Modify the part program.

Program continuation by

14185 Channel %1 Block %2 D number not defined

Explanation
%1 = channel number
%2 = block number, label

The specified H number is not assigned a tool (language mode ISO M).

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14190 Channel %1 block %2 H number with G49

Explanation
%1 = channel number
%2 = block number, label

Both G49 (selection of tool length compensation) and H word unequal to H0 are programmed.

Reaction
Alarm display
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14195 Channel %1 block %2 D number with G49

Explanation
%1 = channel number
%2 = block number, label

Both G49 (selection of tool length compensation) and D word unequal to D0 are programmed.

Reaction
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14197 Channel %1 block %2 Both the D and the H number are programmed

Explanation
%1 = channel number
%2 = block number, label

Both the D and the H words are programmed.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy
Modify the part program.

Program continuation by

14198 Channel %1 block %2 Illegal change of tool direction with tool offset

Explanation
%1 = channel number
%2 = block number, label

If an offset is active in the tool direction, no block may be loaded with which the assignment of the offset axis to the channel axes changes (plane change, tool change milling tool <=> turning tool, geometry axis change)

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited
Reorganize also the correction block.
NC Stop at alarm at block end.

Remedy
- Modify the part program.
- Reduce the offset in the tool direction to zero.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14199

Channel %1 block %2 Illegal plane change when working with tool with diameter component

Explanation
%1 = channel number
%2 = block number, label

If a tool possesses a wear or a length component which is evaluated as a diameter value (bit 0 and / or bit 1 is set in MD $MC_TOOL_PARAMETER_DEF_MASK) and bit 2 is additionally set in this MD, then the tool concerned may only be used in the plane which is active when selecting the tool. A plane change will result in an alarm.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
- Modify the part program.
- Reset bit 2 in MD $MC_TOOL_PARAMETER_DEF_MASK.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14200

Channel %1 block %2 Negative polar radius

Explanation
%1 = channel number
%2 = block number, label

When specifying the end point of a traversing block with G00, G01, G02 or G03 using polar coordinates, the polar radius specified at vocabulary word **RP=...** is negative.

Definition:

- **Specification of the end-of-block point** with *pole angle* and *pole radius* with reference to the current pole (G functions: G00/G01/G02/G03).
- **Redefinition of the pole** with *pole angle* and *pole* radius with reference to the reference point selected using the G function.
  G110 ... last programmed point of the plane
  G111 ... zero point of the current WCS
  G112 ... last pole

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the NC part program accordingly - admissible inputs for the polar radius are only positive, absolute values that specify the distance between the current pole and the block end point (the direction is specified using the polar angle **AP=...**).

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14210 Channel %1 block %2 Pole angle too large

Explanation

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

When specifying the end point of a traversing block with G00, G01, G02 or G03 using polar coordinates, the traversing range of the polar angle has been exceeded, which is programmed at the vocabulary word **AP=...**. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.

Definition:

- **Specification of the end-of-block point** with *pole angle* and *pole radius* with reference to the current pole (G functions: G00/G01/G02/G03).
- **Redefinition of the pole** with *pole angle* and *pole* radius with reference to the reference point selected using the G function.
  G110 ... to the last programmed point of the plane
  G111 ... to the zero point of the current workpiece coordinate system (WCS)
  G112 ... to the last pole

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Correct the NC part program accordingly - the admissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14250 Channel %1 block %2 Negative pole radius

Explanation

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

When redefining the pole with G110, G111 or G112 using polar coordinates, the pole radius specified at the vocabulary word **RP=...** is negative. Only positive, absolute values are admitted.

Definition:

- **Specification of the end-of-block point** with *pole angle* and *pole radius* with reference to the current pole (G functions: G00/G01/G02/G03).
- **Redefinition of the pole** with *pole angle* and *pole radius* with reference to the reference point selected using the G function.
  G110 ... last programmed point of the plane
  G111 ... zero point of the current WCS
  G112 ... last pole

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Correct the NC part program accordingly - admissible 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 specified using the pole angle **AP=...**).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14260 Channel %1 block %2 Pole angle too large

Explanation

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

When redefining the pole with G110, G111 or G112 using polar coordinates, the range of values of the pole angle has been exceeded, which is specified at the vocabulary word **AP=...**. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.

Definition:

- **Specification of the end-of-block point** with *pole angle* and *pole radius* with reference to the current pole (G functions: G00/G01/G02/G03).
- **Redefinition of the pole** with *pole angle* and *pole* radius with reference to the reference point selected using the G function.
  G110 ... last programmed point of the plane
  G111 ... zero point of the current WCS
  G112 ... last pole

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Correct the NC part program accordingly - the admissible input range for the pole angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14270 Channel %1 block %2 Pole incorrectly programmed

Explanation

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

When defining the pole, an axis has been programmed, which is not part of the selected machining plane.

The programming using polar coordinates always refers to the plane enabled using G17 to G19. This also applies to the definition of a new pole with G110, G111 or G112.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Correct the NC part program accordingly - only the two geometry axes forming the current processing plane may be programmed.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14280 Channel %1 block %2 Polar coordinates incorrectly programmed

Explanation

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

The end point of the displayed block was programmed both in the polar coordinate system (with AP=..., RP=...) and in the Cartesian coordinate system (axis addresses X, Y, ...).

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Correct the NC part program accordingly - the axis movement may **only** be specified **in a coordinate system**.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14400

Channel %1 block %2 Tool radius compensation active when switching the transformation

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

The transformation may only be switched with the tool radius compensation active.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy: Use **G40** (in a block with G00 or G01) to carry out a tool radius compensation in the NC part program prior switching the transformation.

Program continuation by: Press NC START to cancel the alarm and continue the program.

14401

Channel %1 block %2 Transformation not existing

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

The desired transformation is not available.

Example:
The following was programmed: N220 TRACYLI(3) ;transformation. no. 3-ON
But, only the transformations 1 and 2 exist.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service.

- Modify the part program, but program only defined transformations.
- Check MD 24100 TRAFO_TYPE_n (assigns the transformation to the appropriate part program statement).

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14404

Channel %1 block %2 Illegal parameterization for transformation

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

An error has occurred when selecting the transformation.

Possible error causes are generally:

- An axis traversed by the transformation is not released:
- because it is occupied by another channel (-> enable the axis)
- is in the Spindle mode (-> enable the axis via SPOS)
- is in the POSA mode (-> enable it via WAITP)
- is a concurring POS axis (-> enable it via WAITP)
- The parameterization via machine data is faulty.

- Axis or geometry axis assignment to the transformation is faulty,
- machine data is faulty (-> change machine data, cold restart)

Please note: Axes not enabled are not reported via EXINAL_TRANSFORM_PARAMETER = 14404, but via EXINAL_ILLEGAL_AXIS = 14092 or BSAL_SYSERRCHAN_RESET = 1011.

Transformation-depending error causes may be:
TRAORI: -

TRANSMIT:

- The current machine axis position cannot be used for selection (e.g. selection in the pole) (-> change the position slightly)
- The parameterization via machine data is faulty.
- Particular prerequisite at machine axis not fulfilled (e.g. rotary axis is no modulo axis) (-> change machine data, cold restart)

TRACYL:

- The programmed parameter is not permitted when selecting a transformation.

Only with the compile cycle "OEM transformation" active:

- The axes involved in the transformation must be referenced!

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy: Please inform the authorized personnel/customer service.
Either modify the part program or change the machine data.

Only with the compile cycle "OEM transformation" active:
Before selecting the transformation, first reference the axes involved in the transformation.

Program continuation by: Press NC START to cancel the alarm and continue the program.

14500

Channel %1 block %2 Illegal DEF or PROC statement in the part program

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

NC part programs with high-level language elements are divided into a **definition part** which is written first and is followed by a **program part.** The transition is not specially marked - no definition statement may follow after the 1st program command.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy: Put the definition and PROC statements to the beginning of the program.

Program continuation by: Press NC START to cancel the alarm and continue the program.

14510

Channel %1 block %2 PROC statement missing in subroutine call

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

When calling a subroutine with parameter transfer ("call-by-value" or "call-by-reference"), the called subroutine must start with a PROC statement.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy
Carry out the definition in the subroutine according to the type you are using.

1. Conventional subroutine structure (without parameter transfer):
   % SPF 123456
   :
   M17
2. Designing the subroutine with vocabulary word and subroutine name (without parameter transfer):
   PROC UPNAME
   :
   M17
   ENDPROC
3. Designing the subroutine with vocabulary word and subroutine name (with "call-by-reference" parameter transfer):
   PROC UPNAME (VARNAME1, VARNAME2, ...)
   :
   M17
   ENDPROC
4. Designing the subroutine with vocabulary word and subroutine name (with "call-by-reference" parameter transfer):
   PROC UPNAME (Typ1 VARNAME1, Typ2 VARNAME2, ...)
   :
   M17
   ENDPROC

Program continuation by
Press NC START to cancel the alarm and continue the program.

14520 Channel %1 block %2 Illegal DEF or PROC statement in the definition program

Explanation
%1 = channel number
%2 = block number, label

The PROC statement may only be written in the beginning of a subroutine.

Reaction
Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy
Modify the NC part program accordingly.

Program continuation by
Press NC START to cancel the alarm and continue the program.

14530 Channel %1 block %2 EXTERN and PROC statements do not match

Explanation
%1 = channel number
%2 = block number, label

Subroutines with parameter transfer must be known **before** they are called in the program. If the subroutines always exist (fixed cycles), the call interfaces are determined by the control system when booting. Otherwise, an EXTERN statement must be programmed in the calling program.

Example:
N123 **EXTERN** UPNAME (TYP1, TYP2, TYP3, ...)

It is imperative that the type of the variable matches or is compatible with the type specified in the definition (PROC statement); the name can be different.

Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Check whether the variable types of the EXTERN and of the PROC statements match with each other and correct them if necessary.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14600 Channel %1 block %2 Cannot create reloading buffer

Explanation	%1 = channel number %2 = block number, label When loading the INITIAL_INI block, the reload buffer could not be created because of insufficient memory in the RAM of the NC.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Provide for free memory space in the NC area, for example, by deleting part programs no longer used.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14601 Channel %1 block %2 Reload buffer could not be cleared

Explanation	%1 = channel number %2 = block number, label The reload buffer for "Execution from external" could not be cleared; probable cause: - HMI-PLC communication not completed.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	All reload buffers are cleared on Power On.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14700 Channel %1 block %2 Timeout at command to interpreter

Explanation	%1 = channel number %2 = block number, label The runtime has exceeded in case of control system-internal commands, such as part program selection, Reset or a modification to configuration-specific machine data.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Please inform the authorized personnel/customer service. If the runtime error occurred due to a load on the system currently too large (e.g. in the HMI area), it is possible that the program/operation will be carried out correctly if it is repeated. Otherwise, contact the A&D system administration, describing the error situation as exact as possible, using the following address:

Siemens AG, System Administration for A&D MC Products, hotline
(tel.: see p. 1-9)

Program continuation by: Power ON

14701 Channel %1 block %2 Number of available NC blocks reduced by %3

Explanation:
%1 = channel number
%2 = block number, label
%3 = number of blocks not available

After Reset, it was detected that the number of available blocks has been reduced in comparison with the last RESET. The cause is a system error. Acknowledge this alarm to continue the execution of the part program.

Reaction:
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: The same procedure as with system errors.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14710 Channel %1 block %2 Error in initialization sequence with function %3

Explanation:
%1 = channel number
%2 = block number, label
%3 = index for section

When the control system boots and is reset, initialization blocks are generated. Due to incorrect machine data settings, errors might occur.

Parameter %3 indicates in which section of ini block generation the error occurred:

Section 0: Error synchronizing leader/main run
Section 1: Error selecting the tool length compensation
Section 2: Error selecting the transformation
Section 3: Error selecting the zero offset

During booting, the cycle interfaces are additionally read in. If an error occurs during this process, "Section 5" is signaled.

Reaction:
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy:
With sections 0 - 3: Load standard machine data
With section 5: Reload cycles

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

14750 Channel %1 block %2 Too many auxiliary functions programmed

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

More than 10 auxiliary functions have been programmed in a block.

Reaction:
Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy
Check whether all auxiliary functions are necessary in the block; modal functions need not be repeated. Either create your own auxiliary function block or distribute the auxiliary functions over several blocks.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14760 Channel %1 block %2 Auxiliary function of a group programmed repeatedly

Explanation
%1 = channel number
%2 = block number, label

The M and H functions can be divided into groups via machine data completely variably if required. Auxiliary functions are summarized in groups such that several individual functions of a group mutually exclude. Within a group, only one auxiliary function is reasonable and permitted.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Please inform the authorized personnel/customer service.
Program only one auxiliary function or auxiliary function group. (For group division see Programming Instructions of the machine manufacturer.)

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14762 Channel %1 block %2 Too many PLC variables programmed

Explanation
%1=channel number
%2 = block number, label

The number of the programmed PLC variables in auxiliary functions has programmed the maximum permissible number; the number is defined via the MD 28150 MM_NUM_VDIVAR_ELEMENTS.

Reaction
Alarm display
Interface signals are set
Interpreter stop
NC Start inhibited
NC Stop at alarm

Remedy
Either modify the part program or change the machine data.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14770 Channel %1 block %2 Auxiliary function incorrectly programmed

Explanation
%1 = channel number
%2 = block number, label

The permissible number of 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).

With the user-defined auxiliary functions, the maximum number of auxiliary functions per group is defined for all auxiliary functions in the NC system settings using machine data 11100 **AUXFU_MAXNUM_GROUP_ASSIGN** (default value: 1).

For each user-defined auxiliary function that is to be assigned a group, the assignment is made using 4 machine data.

MD22010 **AUXFU_ASSIGN_TYPE:** **Auxiliary function type**, e.g. M
MD 22000 **AUXFU_ASSIGN_GROUP:** Desired **group**
MD 22020 **AUXFU_ASSIGN_EXTENSION:** an **extension** (if required)
MD 22030 **AUXFU_ASSIGN_VALUE:** **Function value**

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the part program - max. 16 auxiliary functions, max. 5 M functions per NC block, max. 1 auxiliary function per group.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14780 Channel %1 block %2 Unreleased option used

Explanation
%1 = channel number
%2 = block number, label

An option not released is used in the block.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Modify the part program; acquire the option.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14800 Channel %1 block %2 Programmed path velocity less than or equal to zero

Explanation
%1 = channel number
%2 = block number, label

A negative F value has been programmed in conjunction with the G functions G94, G95 or G96. The path velocity may be programmed in the range from 0.001 to 999 999.999 [mm/min, mm/rev, degrees/min, degrees/rev] for the metric input system and from 0.000 1 to 39 999.999 9 [inch/min, inch/rev] for the inch input system.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Program the path velocity (geometrical sum of the velocity components of the geometry axes involved) within the limits specified above.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

14810 Channel %1 block %2 Negative axis velocity programmed for positioning axis %3

Explanation
%1 = channel number
%2 = block number, label
%3 = axis

A negative feedrate (FA value) was programmed for the axis currently running as a positioning axis. The positioning velocity may be programmed in the range from 0.001 to 999 999.999 [mm/min, mm/rev, degrees/min, degrees/rev] for the metric input system and from 0.000 1 to 39 999.999 9 [inch/min, inch/rev] for the inch input system.

Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Program the positioning velocity within the limits specified above.
Program continuation by	Press NC START to cancel the alarm and continue the program.

14811 Channel %1 block %2 Invalid range of values for acceleration of axis/spindle %3

Explanation	%1 = channel number %2 = block number, label %3 = axis, spindle The permissible input range for the programmable acceleration was not observed; possible values are between 1 and 200 percent.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Adapt the range of values as specified in the Programming Guide.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14815 Channel %1 block %2 Negative thread pitch change programmed

Explanation	%1 = channel number %2 = block number, label A negative thread pitch was programmed.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Correct the value assignment. The programmed F value should be greater than zero. Zero is permissible, but without effect.
Program continuation by	Press NC START to cancel the alarm and continue the program.

14820 Channel %1 block %2 Maximum spindle speed for the constant cutting speed programmed with a negative value

Explanation	%1 = channel number %2 = block number, label You can program a maximum spindle speed for the function "Constant cutting rate G96" using the key word **LIMS=...** The value range is between 0.1 - 999 999.9 [rev/min].
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Program the maximum spindle speed for the constant cutting speed within the limits specified above. The key word LIMS is modal and can be written either in front of the block with the selection of the constant cutting rate or in the block.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

14840 Channel %1 block %2 Value for constant cutting rate out of range

Explanation

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

The programmed cutting rate is out of the input range.

Metric input range:	0.01 to 9 999.99 [m/min]
Inch input range:	0.1 to 99 999.99 [inch/min]

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Program the cutting rate under address S within the permissible value range.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14900 Channel %1 block %2 Both center and end points programmed

Explanation

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

When the circle was programmed using the aperture angle, the circle center point and, in addition, the circle end point has also been programmed. The circle is thus overdetermined. Only one of the two points may be programmed.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Select the programming variant at which the dimensions can be accepted from the workpiece drawing reliably (in order to avoid errors in the calculation).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14910 Channel %1 block %2 Invalid circle aperture angle

Explanation

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

When programming the circle using the opening angle, a negative aperture angle or an aperture angle >= 360 degrees has been programmed.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Program the aperture angle within the permitted value range of 0.0001 - 359.9999 [degrees].

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

14920 Channel %1 block %2 Intermediate point of circle incorrect

Explanation

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

When programming a circle via an intermediate point, all 3 points (start, end and intermediate point) are on a straight line, and the intermediate point (programmed by the interpolation parameters I, J, K) is not between the start and the end point.

Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Either place the position of the intermediate point with the parameters I, J and K such that its position will be really between the circle start and the end point, or sacrifice of this kind of circle programming and program the circle with radius and opening angle or center point parameters.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15030 Channel %1 block %2 Different unit system settings

Explanation	%1 = channel number %2 = block number, label The INCH or METRIC statement describes the unit system where the data records have been read from the control system. To prevent that data that have been intended only for a certain unit system are not incorrectly interpreted, a data record will be supposed if there is only one match is assumed between the above mentioned statement and the currently active unit system setting.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Either change the unit system or load the data record matching with the unit system settings.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15100 Channel %1 block %2 REORG abortion due to log file overflow

Explanation	%1 = channel number %2 = block number, label For synchronization between executing program leader and main run using REORG, the control system requires modification data that are managed in a log file. The alarm displays that no more logfile capacity is available in the channel for the block mentioned above.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Please inform the authorized personnel/customer service. No remedy is possible for further execution of the current part program, but it is possible to: reduce the logfile requirements by: reducing the distance between leader and main run by appropriate leader stops **STOPRE**.
Program continuation by	Use the RESET key to cancel the alarm. Restart the modified (as specified under "Remedy") part program.

15110 Channel %1 block %2 REORG not possible

Explanation

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

For synchronization between executing program leader and main run using REORG, the control system requires modification data that are managed in a log file. The alarm displays that no more logfile capacity is available in the channel for the block mentioned above.

The alarm message indicates that the **log file has been deleted** to provide more additional memory for preparing the program. REORG of the leader memory to the next coincidence no longer possible.

Reaction

Alarm display.

Remedy

Please inform the authorized personnel/customer service.
No remedy is possible for further execution of the current part program, but it is possible to:

reduce the logfile requirements by:
reducing the distance between leader and main run by appropriate leader stops **STOPRE**.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

15150 Channel %1 block %2 Reloading from external source was canceled

Explanation

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

Execution from external source was canceled because the reloading buffer contains not a sufficient number of machine function blocks (traversing blocks, auxiliary function, dwell time etc.). Background: Releasing of the machine function blocks already executed will release memory capacity in the reloading buffer. If no more machine function blocks are released, nothing can be reloaded any more - and no deadlock situation will result.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Insert machine function blocks into the part program.

- Enlarge the reloading buffer ($MN_MM_EXT_PROG_BUFFER_SIZE).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

15160 Channel %1 block %2 Leader configured incorrectly

Explanation

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

The following deadlock was detected in the interpreter:
A block element is needed, the block element memory, however, is empty, and there is no chance to get new block elements by processing the leader/main run queue, since this queue is also empty.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Please inform the authorized personnel/customer service.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

15170

Channel %1 block %2 program %3 could not be compiled

Explanation: %1 = channel number
%2 = block number, label
%3 = string

An error occurred in the compilation mode. The (compiler) message displayed after the error refers to the program specified here.

Reaction: Alarm display.

Remedy: Correct the part program.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

15175

Channel %1 block %2 program %3 No interfaces could be created

Explanation: %1 = channel number
%2 = block number, label
%3 = string

An error has occurred during the interface creation mode. The (compiler) message displayed after the error refers to the program specified here.

Reaction: Alarm display.

Remedy: Correct the part program.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

15180

Channel %1 block %2 Program %3 could not be processed as an INI file

Explanation: %1 = channel number
%2 = block number, label
%3 = string

Errors occurred reading in data as an INI file. The displayed error message refers to the program specified here.

Reaction: Alarm display.

Remedy: Correct the part program.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

15185

Channel %1 %2 Error in INI file

Explanation: %1 = channel number
%2 = number of detected errors

Errors have been detected when executing an INI file.

Reaction: Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service. Correct the INI file or correct the MD and create a new INI file (via 'upload').

Program continuation by: Power ON

15190 Channel %1 block %2 No free memory for subroutine call

Explanation

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

The following deadlock was detected in the interpreter:
Memory for calling the subroutine is required. The module memory, however, is empty, and there is no chance that the module memory becomes free by processing the leader/main run queue, since this queue is also empty.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Program preprocessing stop STOPRE before calling the subroutine.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

15300 Channel %1 block %2 Invalid number of block repetitions with block search

Explanation

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

A negative number of passes has been input in the P column (number of passes) when the function "Block search with calculation" was executed. The permissible value range is P 1 - P 9 999.

Reaction

Alarm display.

Remedy

Enter only positive numbers of passes within the value range.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

15320 Channel %1 block %2 Illegal block search job

Explanation

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

The block search job (type of the search destination) is less than 1 or greater than 5. It is entered in the **Type** column of the search window. Permissible search jobs are:

Type	Meaning
1	Find block number
2	Find label
3	Find string
4	Find program name
5	Find line number in file

Reaction

Alarm display.

Remedy

Alter search job.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

15330 Channel %1 block %2 Illegal block number as a search target

Explanation

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

Syntax error! Only positive integer numbers are permitted as block numbers. With main blocks, a ":" and with auxiliary blocks, an "N" must be put before the block number.

Reaction	Alarm display.
Remedy	Repeat your entry with the corrected block number.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

15340 Channel %1 block %2 Illegal label as a search target

Explanation	%1 = channel number %2 = block number, label Syntax error! A label must contain at least 2 and not more than 32 characters whereby the first two characters must be letters or underscores. Labels must end in a colon.
Reaction	Alarm display.
Remedy	Repeat your entry with the corrected label.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

15350 Channel %1 block %2 Search target not found

Explanation	%1 = channel number %2 = block number, label The specified program has been browsed up to the end of the program without finding the preselected search target.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Check the part program, alter the search target (type error in part program) and restart search.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15370 Channel %1 search target not found with block search

Explanation	%1=channel number An invalid search target (e.g. negative block number) has been specified for block search.
Reaction	Alarm display.
Remedy	Check specified block number, label or string. Repeat your entry with the correct search target.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

15380 Channel %1 block %2 Illegal incremental programming in axis %3

Explanation

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

%3 = axis

After "Block search at the end-of-block point", the transformation has changed. The position acquired during the block search cannot be traversed by increments.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

- Select a search target where the axes are programmed using absolute dimensions.
- Disable adding of the acquired block search position via $SC_TARGET_BLOCK_INCR_PROG = FALSE.
- Use block search with calculation "at the contour".

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

15400 Channel %1 block %2 Selected initial ini block does not exist

Explanation

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

The operator has selected an INI block for a read, write or processing function, which:

1. does not exist in the NCK area, or
2. which does not have the required protection level required to execute the function.

Reaction

Alarm display.

Remedy

Please inform the authorized personnel/customer service.
Check whether the selected INI block is stored in the file system of the NCK.
The protection level must be at least the same (or higher) than the protection level defined when creating the file for the read, write or processing function.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

15410 Channel %1 block %2 Initialization file contains illegal M function

Explanation

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

The only M function permitted in an Init block is the end of program with M02, M17 or M30.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Remove all M functions from the Init block (except for the end-of-block character).

Init blocks may only contain value assignments (and global data definitions if not defined once more in a program executed later), but no movement or synchronous actions.

Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15420 Channel %1 block %2 Instruction not accepted in current mode

Explanation	%1 = channel number %2 = block number, label When executing an Init block, the interpreter has found an illegal instruction (e.g. traversing instruction).
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Remove all movement functions and auxiliary functions from the init block (except for the end-of-block character). Init blocks may only contain value assignments (and global data definitions if not defined once more in a program executed later), but no movement or synchronous actions.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15450 Channel %1 block %2 Compiled program could not be saved

Explanation	%1 = channel number %2 = block number, label A compiled program could not be saved in the compilation mode. This may have one of the following reasons: - Lack of memory capacity - Intermediate code line (Compilat) too long
Reaction	Alarm display.
Remedy	Provide for space in the user memory or modify the part program (not so complex).
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

15460 Channel %1 block %2 syntax conflict with modal G functions

Explanation	%1 = channel number %2 = block number, label The addresses programmed in the block are not compatible with the modal, syntax-defining G function. **Example:N**100 G01 ... I .. J.. K.. LF
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Correct the displayed block; match G functions with addresses in the block with each other.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15700 Channel %1 block %2 Illegal cycle alarm number

Explanation
%1 = channel number
%2 = block number, label

A SETAL command with a cycle alarm number less than 60 000 or greater than 67 999 has been programmed.

Alarm reaction of SIEMENS standard cycles:
No.'s 61 000 - 61 999: Interpreter stop; deletion with Reset
No.'s compensation block; use "Cancel" to cancel the alarm.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Program the alarm number in the SETAL instruction in the right range.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

15800 Channel %1 block %2 Wrong starting conditions for CONTPRON

Explanation
%1 = channel number
%2 = block number, label

The start conditions for contour preparation (CYCLE 95) are faulty:

G40 (deselection of tool radius compensation) is not active

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Modify the part program: Deselect tool radius compensation with G40.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

15810 Channel %1 block %2 Incorrect array dimension with CONTPRON

Explanation
%1 = channel number
%2 = block number, label

The number of columns in a contour table is a fixed value. For the current value, refer to the current Technology Programming Instructions.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the field definition for the contour table.

The number of rows can be freely defined and corresponds to the number of contour elements (circles, straight lines). The number of columns is fixed (as per 6/94: number of columns = 11).

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

15900 Channel %1 block %2 Illegal sensor

Explanation
%1 = channel number
%2 = block number, label

A sensor not existing was selected in the part program for measuring with deletion of the distance to go.

Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the part program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15910 Channel %1 block %2 Illegal sensor

Explanation	%1 = channel number %2 = block number, label A sensor not existing was selected in the part program for measuring without deletion of the distance to go.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the part program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15950 Channel %1 block %2 No traversing motion programmed

Explanation	%1 = channel number %2 = block number, label Measuring with deletion of the distance to go was selected in the part program, without specifying the axis, or the programmed traversing motion is zero.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the part program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

15960 Channel %1 block %2 No traversing motion programmed

Explanation	%1 = channel number %2 = block number, label Measuring without deletion of the distance to go was selected in the part program, without specifying the axis, or the programmed traversing motion is zero.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the part program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16020 Channel %1 in block %2 cannot be repositioned.

Explanation	%1 = channel number %2 = block number, label Faulty programming or operation: A block is to be approached, for which no re-approach information is provided.
Reaction	Alarm display. Interface signals are set.

Interpreter stop.
NC Start inhibited.

Remedy: Modify the part program if necessary.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16100 Channel %1 block %2 axis %3 does not exist in the channel

Explanation: %1 = channel number
%2 = block number, label
%3 = string

Faulty programming:
The spindle number is not known in this channel.
The alarm may occur in conjunction with dwell time or a spindle function.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service.
Check the **part program** to make sure that the programmed spindle number is correct or the program runs in the right channel.

Check MD 35000 **SPIND_ASSIGN_TO MACHAX** for all machine axes to find out whether one of them contains the programmed spindle number. This machine number must be entered in a channel axis of MD 20070 **AXCONF_MACHAX_USED**.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16410 Channel %1 block %2 axis %3 is no geometry axis

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

A geometry axis was programmed, to which no machine axis can be mapped.

Example: Polar coordinate system with X, Z and C axes

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Do not program the appropriate axis as a geometry axis.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16420 Channel %1 block %2 axis %3 programmed repeatedly

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

It is not allowed to program an axis repeatedly.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Delete addresses programmed repeatedly.

Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16440 Channel %1 block %2 Rotation programmed for geometry axis that does not exist

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number A rotation was programmed, which will rotate a geometry axis that does not exist.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the part program.
Program continuation by	Press NC START to cancel the alarm and continue the program.

16500 Channel %1 block %2 Chamfer or radius negative

Explanation	%1 = channel number %2 = block number, label A negative chamfer or rounding has been programmed under the vocabulary words CHF= .., RND=...
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Use only positive values to program chamfers, roundings and modal roundings.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16510 Channel %1 block %2 No transversal axis existing

Explanation	%1 = channel number %2 = block number, label The diameter programming was activated using the vocabulary word DIAMON although no transversal axis is programmed in this NC block. If the diameter axis is no geometry axis, with the default setting DIAMON, the alarm will already occur when the control system is turned.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Please inform the authorized personnel/customer service. Activate the modally active G function only in NC blocks which contain a transversal axis or disable the diameter programming via DIAMOF. Select DIAMOF for the default setting in the machine data 20150 **GCODE_RESET_VALUES[28]**.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16700 Channel %1 block %2 axis %3 Invalid feed type

Explanation

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The feed for a thread cutting function has been programmed with an invalid unit.

1. **G33 (thread with constant pitch)** and feed have not been programmed with G94 or G95.
2. **G33 (thread with constant pitch)** is active (maintained) and, in addition, G63 is programmed in a following block → conflict! (G63 is in the 2nd, and G33, G331, as well as G332 are in the 1st G group).
3. **G331** or **G332 (rigid tapping)** and feed have not been programmed with G94.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Use only feed type G94 or G95 for thread cutting functions.

Use G01 to deselect the thread cutting function after G33 and before G63.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

16715 Channel %1 block %2 axis %3 Master spindle not at a standstill

Explanation

%1 = channel number
%2 = block number, label
%3 = spindle number

The spindle must be at a standstill when the function is in use (G74, reference point approach).

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Program M5 or SPOS in the part program before the faulty block.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

16720 Channel %1 block %2 axis %3 Thread pitch is zero

Explanation

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

In a thread block with **G33** (thread with constant lead) or G331 (rigid tapping), no pitch was programmed.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

The thread lead for the specified geometry axis must be programmed with the corresponding interpolation parameter.

X → I
Y → **J**
Z → **K**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16730

Channel %1 block %2 axis %3 Wrong parameter(s)

Explanation:

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

With **G33** (thread cutting with constant lead), the lead parameter has not been assigned to the axis defining the velocity.

With longitudinal and transversal threads, the thread lead for the specified geometry axis is programmed with the corresponding interpolation parameter.

X → I
Y → **J**
Z → **K**

With tapered threads, the address **I, J, K** depends on the address with the larger path (thread length). A 2nd pitch for the other axis, however, will **not be specified.**

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Assign the pitch parameters of the axis defining the velocity.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16740

Channel %1 block %2 No geometry axis programmed

Explanation:

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

With thread cutting (G33) or rigid tapping (G331, G332), no geometry axis was programmed. The geometry axis, however, is absolutely necessary if an interpolation parameter was specified.

Example:

```
N100 G33 Z400 K2          ; Pitch 2 mm, end of
  :                         thread Z=400 mm
N200 SPOS=0     ; Switch spindle to axis mode
N201 G90 G331 Z-50 K-2 ; Tapping to Z=-50, CCW rotation
N202 G332 Z5    ; Retraction, automatic direction reversal
N203 S500 M03  ; Spindle back to spindle mode
```

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Specify a geometry axis and the appropriate interpolation parameter.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16750

Channel %1 block %2 axis %3 SPCON not programmed

Explanation:

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The programmed function (rotary axis, positioning axis) requires the spindle to be in the position control mode.

Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Program the position control of the spindle with SPCON in the previous block.
Program continuation by	Press NC START to cancel the alarm and continue the program.

16751 Channel %1 block %2 spindle/axis %3 SPCOF cannot be executed

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number The programmed function requires the spindle to be in the control mode. Do not deselect the position control in the positioning or axis mode.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	In the previous block, switch the spindle to the control mode. This can be done using M3, M4 or M5 for the spindle concerned.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16760 Channel %1 block %2 axis %3 S value missing

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number The spindle speed for rigid tapping chuck (G331 or G332) has not been specified.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Program the spindle speed under address S in [r.p.m.] (despite the axis mode); the direction of rotation results from the sign of the spindle pitch. Positive pitch: Direction of rotation as with M03 Negative pitch: Direction of rotation as with M04
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

16762 Channel %1 block %2 spindle %3 Thread function active

Explanation	%1 = channel number %2 = block number, label %3 = spindle number Faulty programming: The spindle function cannot be executed at the moment. The alarm occurs if the spindle is linked with axes via interpolation.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Modify the part program. Deselect thread cutting or tapping.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16763 Channel %1 block %2 axis %3 Illegal speed programmed (zero or negative)

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

A speed (S value) with zero value or negative value has been programmed.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: The programmed speed (S value) must be positive. Depending on the particular application, zero can also be accepted (e.g. G25 S0).

Program continuation by: Use the RESET key to cancel the alarm.

16770 Channel %1 block %2 axis %3 No measuring system installed

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

A position was programmed for the axis which requires a measuring system. Acc. to MD 30 200 **NUM_ENCS**, this machine axis has no measuring system.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Remove the appropriate function (e.g. SPOS) from the part program or enter an existing measuring system in MD 30 200 **NUM_ENCS.**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16800 Channel %1 block %2 Traversing instruction DC/CDC not allowed for axis %3

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The vocabulary word DC (**D**irect **C**oordinate) may only be applied to rotary axes. It results in approaching the programmed **absolute position** using the shortest way.

Example:N100 C=DC(315)

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service.
Replace the vocabulary word DC in the displayed NC block by **AC** (**A**bsolute **C**oordinate).

If the alarm display is based on an incorrect axis definition, the axis can be declared a rotary axis using the axis-specific machine data MD 30300 **IS_ROT_AX**.

Related machine data:

MD 30310 **ROT_IS_MODULO**
MD 30320 **DISPLAY_IS_MODULO**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16810 Channel %1 block %2 Traversing instruction ACP not allowed for axis %3

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The vocabulary word **ACP** (**A**bsolute **C**oordinate **P**ositive) are only allowed for modulo axes. It results in approaching the programmed **absolute position** in the specified direction.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service.
Replace the vocabulary word ADC in the displayed NC block by **AC** (**A**bsolute **C**oordinate).

If the alarm display is based on an incorrect axis definition, the axis can be declared a rotary axis with modulo conversion using the axis-specific machine data MD 30300 **IS_ROT_AX** and MD 30310 **ROT_IS_MODULO** .

Related machine data:

MD 30 320 **DISPLAY_IS_MODULO**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16820 Channel %1 block %2 Traversing instruction ACN not allowed for axis %3

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The vocabulary word **ACN** (**A**bsolute **C**oordinate **P**ositive) is only allowed for modulo axes. It results in approaching the programmed **absolute position** in the specified direction.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Please inform the authorized personnel/customer service.
Replace the vocabulary word ADN in the displayed NC block by **AC** (**A**bsolute **C**oordinate).

If the alarm display is based on an incorrect axis definition, the axis can be declared a rotary axis with modulo conversion using the axis-specific machine data MD 30300 **IS_ROT_AX** and MD 30310 **ROT_IS_MODULO** .

Related machine data:

MD 30320 **DISPLAY_IS_MODULO**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

16830 Channel %1 block %2 Invalid position programmed for axis/spindle %3

Explanation
%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

A position for the modulo axis has been programmed out of the range 0 ... 359.999.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Program a position within the range 0 ... 359.999.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

16903 Channel %1 Program control:Action %2 not allowed in the current state

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The action concerned cannot be processed at the moment. This can occur when importing machine data, for example.

Reaction
Alarm display.

Remedy
Wait until the previous process is completed or press RESET to cancel and repeat the operating action.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

16904 Channel %1 Program control:Action %2 not allowed in the current state

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The operation/process in question (program, Jog, block search, reference point, ...) cannot be started or continued in the current state.

Reaction
Alarm display.

Remedy
Check program state and channel state.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

16905 Channel %1 Program control:Action %2 not allowed

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The execution cannot be started or continued. A start will only be accepted if an NCK function can be started.
Example: A start is accepted in Jog mode, for example, if the function generator is active or a Jog movement has been stopped by the Stop key beforehand.

Reaction
Alarm display.

Remedy
Check program state and channel state.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

16906 Channel %1 Program control:Action %2 is canceled because of an active alarm

Explanation	%1 = Channel number %2 = Action number/action name (see Section 1.5 "Action List") The action has been canceled due to an alarm. The execution cannot be started or continued. A start will only be accepted if an NCK function can be started.
Reaction	Alarm display.
Remedy	Eliminate the error and acknowledge the alarm. Then restart the process.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

16907 Channel %1 action %2 only possible in Stop condition

Explanation	%1 = Channel number %2 = Action number/action name (see Section 1.5 "Action List") The action may only be carried out in Stop condition.
Reaction	Alarm display.
Remedy	Check program state and channel state.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

16908 Channel %1 action %2 only possible in reset state or at block end

Explanation	%1 = Channel number %2 = Action number/action name (see Section 1.5 "Action List") The action %2 may only be executed at RESET or at the end of the block.
Reaction	Alarm display.
Remedy	Check program state and channel state.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

16909 Channel %1 action %2 is not permitted in current mode

Explanation	%1 = Channel number %2 = Action number/action name (see Section 1.5 "Action List") Another mode must be activated for the activated function.
Reaction	Alarm display.
Remedy	Check operation and operating mode.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

16911 Channel %1 Switching to another mode is not permitted

Explanation	%1=channel number Switching from Overstore to another operating mode is not permitted.
Reaction	Alarm display.
Remedy	After you have quitted the Overstore mode, you can switch to another mode.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

16912 Channel %1 Program control:Action %2 only possible in RESET state

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

This action can only be carried out in Reset condition.
Example: A program selection by HMI or channel communication (INIT) can only be carried out in the RESET state.

Reaction: Alarm display.

Remedy: RESET or wait until the processing is completed.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16913 Mode group %1 channel %2 Mode switching:Action %3 not permitted

Explanation
%1 = channel number
%2 = mode group number
%3 = action number/action name (see Section 1.5 "Action List")

Switching to the desired mode is not allowed; switching is only allowed in Reset state.

Example:
The program execution is stopped in AUTO mode by NC Stop. Then, a mode change to JOG is carried out program state "interrupted"). From this operating mode, you can change only to AUTO mode, but not to MDA!

Reaction: Alarm display.

Remedy: Either press the Reset key , thus resetting the program execution, or select the mode in which the program execution has been executed until this moment.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16914 Mode group %1 channel %2 Mode switching:Action %3 not permitted

Explanation
%1 = channel number
%2 = mode group number
%3 = action number/action name (see Section 1.5 "Action List")

Illegal mode switching, e.g.: AUTO → MDAREF

Reaction: Alarm display.

Remedy: Check operation or selected mode.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16915 Channel %1 action %2 not permitted in current block

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

If traversing blocks are interrupted by ASUPs, the continuation of the interrupted program (reorganization of block execution) must be possible after the end of the ASUP.
The 2nd parameter describes the action that wanted to interrupt the block execution.

Reaction: Alarm display.

Remedy: Continue program execution up to a reorganized NC block or modify part program.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16916 Channel %1 repositioning:Action %2 not allowed in the current state

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

Repositioning of block execution is currently not possible. Mode change cannot be carried out.
The 2nd parameter described the action recommended to use for repositioning.

Reaction: Alarm display.

Remedy: Continue the program to an NC block which can be repositioned, or modify the part program.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16919 Channel %1 action %2 is not allowed because of an alarm

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The action cannot be executed due to an alarm or the channel is in the error status.

Reaction: Alarm display.

Remedy: Press the RESET key.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16920 Channel %1 action %2 is already active

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

An identical action is still being executed.

Reaction: Alarm display.

Remedy: Wait until the previous operation is completed and then repeat the action.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16922 Channel %1 subroutines:action %2 Maximum nesting depth exceeded

Explanation
%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The current operation can be interrupted by various actions.
Depending on the action, control-internal programs are activated. These programs can be interrupted in similar fashion as the NC program. For memory reasons, random nesting depths of the control-internal programs are not possible.

Example:
The current program execution is interrupted by an interrupt. Any program executions activated prior to this program activation are interrupted by interrupts of a higher priority.
Possible actions are Dry Run, Single Block Decoding, Clear Distance to Go, etc.

Reaction: Alarm display.
Interface signals are set.
NC Start inhibited.
NC Stop at alarm.

Remedy: Press the RESET key.

Before starting the program, check the program nesting depth and avoid interruption.

Example: It is not recommended to interrupt the approach block of a repositioning process.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16923

Channel %1 Program control:Action %2 not allowed in the current state

Explanation: %1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

Current program execution cannot be stopped, as merely a preprocessing process is active.

This applies, e.g. to the loading machine data and block search until the searched target is found.

Reaction: Alarm display.
Interface signals are set.

Remedy: Press Reset to cancel.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16924

Channel %1 caution:Program test will change tool management data

Explanation: %1=channel number

When testing the program, the workpiece data are modified. The data cannot automatically be corrected when the program test is completed.

This error message prompts the operator to make a backup copy of the data and to reload them after the program has been tested.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/customer service.
Save the tool data to HMI and reload/recopy them after "ProgtesOff".

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16925

Channel %1 Program control:Action %2 not allowed in the current state

Explanation: %1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The action has been denied, since a mode change or submode change (switchover to AUTOMATIC, MDA, JOG) is currently carried out.

Example:
This alarm message appears if the Start button is pressed when switching a mode or submode, e.g. from AUTOMATIC to MDA before the NC has confirmed the mode selection.

Reaction: Alarm display.

Remedy: Repeat the action.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16927

Channel %1 action %2 not allowed during active interrupt handling

Explanation: %1=channel number

%2 = action number/action name (see Section 1.5 "Action list")

The action may not be activated when handling an interrupt (e.g., mode switching).

Reaction: Alarm display.

Remedy: RESET or wait until the interrupt processing is completed.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16928

Channel %1 interrupt handling:Action %2<ALNX> not possible

Explanation:

%1 = channel ID
%2 = action number/action name (see Section 1.5 "Action List")

A program interruption to a block that cannot be reorganized has been activated.

Examples of possible program interruption in this case:
- Travel to fixed stop
- Vdi channel: delete distance to go
- Vid axial: delete distance to go
- Measuring
- Software limit
- Axis change
- Axis from follow-up mode
- Servo Disable
- Gear step change: Actual gear step unequal to set gear step

The block concerned is:
a block acquired during the block search (except the last block acquired); block from Overstore Off.

Reaction:

Alarm display.
Interface signals are set.
NC Start inhibited.
NC Stop at alarm.

Remedy: Do no initiate this event in this block.

Program continuation by:

16930

Channel %1:Predecessor and current block %2 must be separated by an executable block

Explanation:

%1=channel number
%2 = block number

The language function MSG must be packed into separate NC blocks. To avoid velocity dips, these blocks will be added to the following NC block internally in the NC (with WAITMC, to the previous NC_block). For this reason, an executable block (no arithmetic block) must always be between the NC blocks.
An executable NC block contains, e.g. traversing movements, an auxiliary function, Stopre, dwell time,....

Reaction:

Alarm display
The interface signals will be set.
Reorganize also the correction block.
NC Stop at alarm.

Remedy: Program an executable NC block between the previous block and the current NC block.

Program continuation by: Press NC START to clear the alarm. Restart the part program.

16931

Channel %1 subroutines:action %2 Maximum nesting depth exceeded

Explanation

%1 = Channel number
%2 = Action number/action name (see Section 1.5 "Action List")

The current operation can be interrupted by various actions.
Depending on the action, internal programs are activated. These can be interrupted in the same manner as the user program. For memory reasons, any nesting depth of these programs is not possible.

Example:
Do not interrupt the approach block of a repositioning process repeatedly, but wait until the approach block is executed.
Possible actions are mode change, SlashON/Off

Reaction

Alarm display.

Remedy

Initiate block change and repeat the action.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

16932

Channel %1 Conflict activating user data of the type %2

Explanation

%1=channel number
%2 = data type

The function "Activate user data" (PI service _N_SETUDT) will modify a data record (tool compensation, settable zero offset or base frame) which is also written by the part program blocks being prepared.

In case of a conflict, the value entered by the MMC is reset.

Parameter %2 specifies the data record concerned:

1: active tool compensation

2: base frame

3: active zero offset

Reaction

Alarm display.

Remedy

Check the inputs on the MMC and repeat them if necessary.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

16934

Channel %1 interrupt handling:Action %2<ALNX> not possible due to STOP

Explanation

%1 = channel ID
%2 = action number/action name (see Section 1.5 "Action List")

Reorg events are, e.g. subroutine cancelation, deletion of the distance to go and interrupts, axis replacement, quitting of the follow-up mode.

In this situation, two Reorg events occur at the same time. The 2nd Reorg event occurs at the same time as the 1st block which was generated by the previous event. (e.g.: Axis replacement is enforced 2x quickly in succession). Axis change in the channels results in Reorg, from which an axis is removed without preparation.

To execute the sequence mentioned above, exactly this block must be stopped so that the Ipo buffer cannot be filled any longer. This can be carried out using the Stop key or by StopAll, by an alarm with configuration of interpreter stop or by decoding single block.

Reaction

Alarm display.
Interface signals are set.

NC Start inhibited.
NC Stop at alarm.

Remedy: Program must be aborted using Reset.

Program continuation by:

16936

Channel %1 action %2<ALNX> not possible due to active dry run feed

Explanation: %1 = channel ID
%2 = action number/action name

The action is not allowed since dry run feed is currently active.

Reaction: Alarm display.

Remedy: Program must be aborted using Reset.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16937

Channel %1 action %2<ALNX> not possible due to program test

Explanation: %1 = channel ID
%2 = action number/action name

The action is not allowed since program test is currently active.

Reaction: Alarm display.

Remedy: Deactivate the program test.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

16938

Channel %1 action %2<ALNX> canceled due to active gear stage change

Explanation: %1 = channel ID
%2 = action number/action name (see Section 1.5 "Action List")

Reorg events are, e.g. subroutine cancellation, deletion of the distance to go and quitting of the follow-up status.

These events wait until the gear stage change is completed. The maximum waiting time, however, has elapsed.

Reaction: Alarm display.
Interface signals are set.
NC Start inhibited.
NC Stop at alarm.

Remedy: Program must be aborted using Reset.

Program continuation by: Use the RESET key to cancel the alarm.

16939

Channel %1 action %2<ALNX> denied due to active gear stage change

Explanation: %1 = channel ID
%2 = action number/action name (see Section 1.5 "Action List")

Reorg events which are possible in the stopped condition, such as mode switching, wait until the gear stage change is completed. The maximum waiting time, however, has elapsed.

Reaction: Alarm display.
Interface signals are set.

Remedy: Repeat the action.

Program continuation by	Use the RESET key to cancel the alarm.

16940

Channel %1 action %2<ALNX> wait for gear stage change

Explanation	%1 = channel ID %2 = action number/action name (see Section 1.5 "Action List") Reorg events wait until the gear stage change is completed. During the waiting time, this alarm is displayed.
Reaction	Alarm display. Display of a warning message.
Remedy	It is a self-clearing note.
Program continuation by	The alarm display will disappear with the alarm cause. No further operation required.

16941

Channel %1 action %2<ALNX> denied since no program event executed

Explanation	%1 = channel ID %2 = action number/action name The setting of the machine data $MC_PROG_EVENT_MASK requires an Asup to be initiated automatically in case of RESET or PowerOn. The Asups initiated implicitly are generally called "Event-controlled program call" or "Program event". In the alarm situation, however, this Asup could not be activated; therefore, the action (usually, the part program start) must be denied. Reasons why the Asup could not be initiated: 1. The Asup program does not exist (/ N CMA DIR/ _N_PROG_EVENT_SPF) 2. READY missing (because of alarm)
Reaction	Alarm display.
Remedy	• Load the program. • Check $MN_ASUP_START_ASK. • Acknowledge the alarm.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

17001

Channel %1 block %2 No more memory for tool/magazine data

Explanation	%1 = channel number %2 = block number, label The number of the tool data in the NC is limited.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Delete tools not used.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

17010 Channel %1 block %2 No more memory available

Explanation
%1 = channel number
%2 = block number, label

When executing/importing files of the active user memory, it was detected that the memory available is not sufficient (e.g. for creating the tool offset memory).

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Make more memory space available for subroutine calls and tool offsets.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17020 Channel %1 block % Illegal array - index1

Explanation
%1 = channel number
%2 = block number, label

A read or write access to a field variable (e.g. arithmetic parameter) with invalid 1st field index has been programmed.

e.g. R2000 = 5; parameter number 2000 not defined

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the specification of the field elements for the access instruction with respect to the defined size.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17030 Channel %1 block %2 Illegal array index2

Explanation
%1 = channel number
%2 = block number, label

A read or write access to a field variable has been programmed with invalid 2nd field index has been programmed. Field index programmed. Valid field indices are within the defined field size and the absolute limits (0 - 32 766).

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the specification of the field elements for the access instruction with respect to the defined size.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17040 Channel %1 block % Illegal axis array - index1

Explanation
%1 = channel number
%2 = block number, label

A read or write access was programmed to an axial variable with which the axis name can no longer unambiguously be mapped to a machine axis.

Reaction
Alarm display.
Interface signals are set.

Interpreter stop.
NC Start inhibited.

Remedy: Use the machine axis name as the axis index.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

17050 Channel %1 block % Illegal value

Explanation:

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

When accessing an individual frame element, a frame component other than TRANS, ROT, SCALE or MIRROR was addressed.

The frame components are either selected using the vocabulary words

TR for the offset (TRANS, internally 0)
RT for the rotation (ROT, internally 1)
SC for the scaling (SCALE, internally 3) and
MI for mirroring (MIRROR, internally 4)

or specified directly as an integer value 0, 1, 3, 4.

Example:
Access to the rotation around the X axis of the frame which can currently be set.

R10=$P_UIFR[2, X, RT] can also be programmed as:
R10=$P_UIFR[2, X, 1]

Reaction:

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Address frame components only using the appropriate vocabulary words; program the scaling factor within the limits of 0.000 01 up to 999.999 99.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

17070 Channel %1 block %2 Data is write-protected

Explanation:

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

You have tried to write into a write-protected variable or an MD, for which you have no right to access.

Reaction:

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: Remove the write accesses to write-protected variables from the NC program or the machine data file.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

17080 Channel %1 block %2 Value less than lower limit

Explanation:

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

You have tried to write into an MD with a value less than the defined lower limit.

Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Please inform the authorized personnel/customer service. Determine the input limits of the MD and carry out value assignment within these limits.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

17090 Channel %1 block %2 Value greater than upper limit

Explanation	%1 = channel number %2 = block number, label You have tried to write into an MD with a value greater than the defined upper limit.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Please inform the authorized personnel/customer service. Determine the input limits of the MD and carry out value assignment within these limits.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

17095 Channel %1 block %2 Invalid value

Explanation	%1 = channel number %2 = block number, label You have tried to write an invalid value, e.g. zero, to a machine data.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Correct the value assignment, e.g. a value within the range of value, which is equal to zero.
Program continuation by	

17160 Channel %1 block %2 No tool selected

Explanation	%1 = channel number %2 = block number, label You have tried to access the current tool compensation data although no tool has been selected.
Reaction	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
Remedy	Program or activate a tool compensation in the NC part program. **Example:**N100 G.. ... T5 D1 ... LF
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

17180 Channel %1 block %2 Illegal D number

Explanation

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

In the displayed block, a D number (edge number) is accessed which is not initialized and therefore does not exist.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Check tool call in NC part program:

- Correct cutting edge number D.. programmed?
  If no edge number is specified, D1 is automatically active.
- All tool parameters defined?
  The dimensions of the tool edge must have been entered beforehand either via the operator panel or the V24 interface.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

17181 Channel %1 block %2 T no.= %3, D no.= %4 does not exist

Explanation

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

A D number has been programmed, which the NC does not know. By default, the D number refers to the specified T number. If the function "Flat D number" is active, T= 1 is output.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy

If the program is not correct, then use the correction block to eliminate the error and continue the program.
If the data record is missing, load a data record for the corresponding T/D into the NC (via HMI, with "Overstore") and then continue the program.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

17188 Channel %1 D number %2 defined with tools no. %3 and %4

Explanation

%1 = channel number
%2 = offset number D
%3 = T number of first tool
%4 = T number of second tool

No unambiguity of the a/m D number %2 in the tool list of channel %1 is guaranteed.
The a/m T numbers %3 and %4 have one compensation each with number %2.

Reaction

Alarm display.
Interface signals are set.

Remedy

1. Provide for unambiguity of the D numbers in the TO unit.
2. If no unambiguity is required in the following, do not use the command in question.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

17190 Channel %1 block %2 Illegal T number

Explanation

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

In the displayed block, a T number (tool number) is accessed which is not initialized and therefore does not exist.

Reaction

Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy

Check tool call in NC part program:

- Correct tool number T... programmed?
- Tool parameters P1 - P25 defined?
  The dimensions of the tool edge must have been entered beforehand either via the operator panel or the V24 interface.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

17191 Channel %1 block %2 T= %3 does not exist, program %4

Explanation

%1 = channel number
%2 = block number, label
%3 = T number or T identifier
%4 = program name

A tool identifier was programmed, which the NC does not know.

Reaction

Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy

If the program pointer is positioned on an NC block that contains the a/m identifier:
If the program is not correct, then use the correction block to eliminate the error and continue the program.
If the data record is missing, create a data record. That means: Load the data record of the tool including all defined D numbers into the NC (via HMI) and continue the program.
If the program pointer is positioned on an NC block that does not contain the a/m T identifier:
The error already occurred earlier in the program when programming T, the alarm, however, is only output with the change command.

If there is an error in the program - T5 programmed instead of T55 -, then you can correct the current block using the correction block; i.e. if only M06 is written, you can correct the current block into T55 M06. The wrong line T5 will remain in the program unless canceled via RESET or end of program.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

17194 Channel %1 block %2 No suitable tool found

Explanation

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

- It was tried to access a tool not defined.
- The specified tool does not allow an access.
- A tool with the desired properties is not available.

Reaction

Alarm display.
Reorganize also the correction block.
Interface signals are set.

Remedy
Check the access to the tool:
- Is the language command parameterized correctly?
- May the tool not permit the access due to its status?

Program continuation by
Press NC START to cancel the alarm and continue the program execution.

17200

Channel %1 block %2: Tool cannot be deleted

Explanation
%1 = channel number
%2 = block number, label

You have tried to remove the tool data for a workpiece currently being machined from the part program. Tool data for workpieces currently being machined must not be deleted.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Deselect tool.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17210

Channel %1 block %2 Variable access not possible

Explanation
%1 = channel number
%2 = block number, label

The variable cannot be read/written directly from the part program.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Modify the part program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17220

Channel %1 block %2 Tool does not exist

Explanation
%1 = channel number
%2 = block number, label

Whenever it is tried to use a T no. to access a tool not (yet) defined.

Reaction
Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy
Correct the NC program.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

17270

Channel %1 block %2 call-by-reference: Illegal variable

Explanation
%1 = channel number
%2 = block number, label

Machine data and system variables may not be transferred as a call-by-reference parameter.

Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Modify the NC program: Either assign the value of the machine data or of the system variable to a program-local variable or transfer it as a parameter.
Program continuation by	Press NC START to cancel the alarm and continue the program.

17610 Channel %1 block %2 Positioning axis %3 cannot take part in the transformation

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number The axis addressed via the vocabulary word POS or POSA is involved in the active transformation. It can therefore not be traversed as a positioning axis.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Remove the POS or POSA statement from the part program block or deselect the transformation first using TRAFOOF.
Program continuation by	Press NC START to cancel the alarm and continue the program.

17620 Channel %1 block %2 Fixed-point approach not possible for transformed axis %3

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number In the displayed block, an axis is programmed for the fixed-point approach (G75) which is involved in the active transformation. Thus, fixed-point approach will not be executed.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Remove the G75 statement from the part program block or deselect the transformation first using TRAFOOF.
Program continuation by	Press NC START to cancel the alarm and continue the program.

17630 Channel %1 block %2 Referencing not possible for transformed axis %3

Explanation	%1 = channel number %2 = block number, label %3 = axis name, spindle number In the displayed block, an axis is reference point approach (G74) is programmed for the fixed-point approach (G75) which is involved in the active transformation. Thus, the reference point approach will not be executed.
Reaction	Alarm display. Interface signals are set. Reorganize also the correction block.
Remedy	Remove the G74 statement or the machine axes involved in the transformation from the part program block or first deselect the transformation using TRAFOOF.

Program continuation by: Press NC START to cancel the alarm and continue the program.

17640 Channel %1 block %2 Spindle mode not possible for transformed axis %3

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The axis programmed for the spindle mode is involved in the current transformation as a geometry axis. This is not permissible.

Reaction: Alarm display.
Interface signals are set.
Interpreter stop.
NC Start inhibited.

Remedy: First deactivate the transformation.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

17650 Channel %1 block %2 Machine axis %3 cannot be programmed

Explanation: %1 = channel number
%2 = block number, label
%3 = axis name, spindle number

the machine axis cannot be used with the transformation active.
Possibly, the function can also be programmed in another coordinate system.
The appropriate axis identifier is used to select the coordinate system.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Remedy: Either disable the transformation or use a different coordinate system.

Program continuation by: Press NC START to cancel the alarm and continue the program.

18100 Channel %1 block %2 FXS[ ] was assigned an illegal value

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

Currently, only the values:

0: "Deselect travel to fixed stop"

1: "Select travel to fixed stop" are valid.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Program continuation by: Press NC START to cancel the alarm and continue the program.

18101 Channel %1 block %2 FXST[ ] was assigned an illegal value

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

Currently, only the range of values 0.0 ... 100.0 is valid.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Program continuation by: Press NC START to cancel the alarm and continue the program.

18102 Channel %1 block %2 FXSWU was assigned an illegal value

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

Currently, only positive values including zero are valid.

Reaction: Alarm display.
Interface signals are set.
Reorganize also the correction block.

Program continuation by: Press NC START to cancel the alarm and continue the program.

18310 Channel %1 block %2 frame:Illegal rotation

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

Rotations not possible with global frames.

Reaction: Alarm display
Interface signals are set
Interpreter stop
NC Start inhibited
NC Stop at alarm

Remedy: Modify the part program.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

18311 Channel %1 block %2 frame:Illegal instruction

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

Reading or writing a frame that does not exist.

Reaction: Alarm display
Interface signals are set
Interpreter stop
NC Start inhibited
NC Stop at alarm

Remedy: Modify the part program.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

18314 Channel %1 block %2 frame:Type conflict

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

It is not possible to link global frames and channel-specific frames.
The alarm also occurs if a global frame is programmed with a channel/axis identifier and no machine axis exists for this channel axis.
Channel-specific frames cannot be programmed using machine axis identifiers if there is no channel axis assigned to the machine axis.

Reaction: Alarm display
Interface signals are set
Interpreter stop
NC Start inhibited
NC Stop at alarm

Remedy: Modify the part program.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

20000

Channel %1 axis %2 Reference cam not reached

Explanation: %1 = channel number
%2 = axis name, spindle number

After starting reference point approach, the rising edge of the reducing cam must be reached within the distance defined in MD 34 030 **REFP_MAX_CAM_DIST** (phase 1 of referencing). (This error occurs **only with incremental encoders**).

Reaction: NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy: Please inform the authorized personnel/customer service.
This may have three possible error causes:

1. The value in MD 34 030 **REFP_MAX_CAM_DIST** is too small.
   Determine the maximum possible path from the beginning of referencing to the reducing cam and compare with the value in MD 34 030**REFP_MAX_ CAM_DIST**; if necessary increase the MD.
2. The cam signal does not reach the PLC input unit.
   Actuate the reference-point switch manually and check the input signal at the NC/PLC interface (Path: switch! plug! Cable!PLC input! user program).
3. The reference-point switch is not actuated by the cam.
   Check the vertical distance between reducing cam and actuating switch.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

20001

Channel %1 axis %2 Cam signal missing

Explanation: %1 = channel number
%2 = axis name, spindle number

At the beginning of phase 2 of reference-point approach, the signal from the reducing cam is no longer available.

Phase 2 of reference-point approach starts when the axis stops on the reducing cam after decelerating. Then the axis starts in the opposite direction to select the next zero mark of the measuring system when leaving/re-approaching (negative/positive edge) the reducing cam.

Reaction: NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy: Check whether the **deceleration distance** from the starting velocity is greater than the reference cam - in this case, the axis can only stop behind the cam.
User longer cams.

When the axis has stopped on the cam, check whether the signal "DELAYED REFERENCE-POINT APPROACH" (V380x1000.7) is still provided.

- Hardware: wire break? short circuit?
- Software: user program?

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

20002 Channel %1 axis %2 Zero mark missing

Explanation

%1 = channel number
%2 = axis name, spindle number

The zero mark of the **incremental position encoder** is not within the defined path.

Phase 2 of reference point approach ends if the zero mark of the encoder has been detected after the rising edge of the PLC interface signal "DELAYED REFERENCE POINT APPROACH"
(V 380x1000.7) has provided trigger start. The maximum distance to be traversed between trigger start and the following zero mark is defined in
MD 34 060 **REFP_MAX_MARKER_DIST**.

The monitoring prevents the zero mark signal from being overtraveled and the next zero mark signal from being evaluated as a reference-point signal! (insufficient cam adjustment or too large delay by the PLC user program).

Reaction

NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Check the cam adjustment and provide for sufficient distance between the end of the cam and the zero mark signal following after the end of the cam. The distance must be longer than the distance which the axis can traverse within the PLC cycle time.

Increase MD 34 060 **REFP_MAX_MARKER_DIST**, but the value may not be higher than the distance between 2 zero marks. This would possibly switch off the monitoring!

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

20004 Channel %1 axis %2 Reference mark missing

Explanation

%1 = channel number
%2 = axis name, spindle number

The required 2 reference marks of the **distance-coded length measuring system** have not been found within the defined search path (axis-specific MD 34 060 **REFP_MAX_MARKER_ DIST).**

Distance-coded scales do not need reducing cams. (But if any, it will be evaluated.) The conventional direction key defines the search direction.
The search path MD 34060 **REFP_MAX_MARKER_ DIST** , within 2 reference marks are expected, counts from the start point.

Reaction

NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Please inform the authorized personnel/customer service.
Determine the **distance** between 2 odd reference marks (reference-point interval). Then enter this value (in the case of Heidenhain scales, 20.00 mm) in MD 34 060 **REFP_MAX_MARKER_DIST**.

Check the **reference track** of the scale incl. evaluation electronics.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

20005 Channel %1 axis %2 Reference point approach was canceled

Explanation

%1 = channel number
%2 = axis name, spindle number

The channel-specific referencing could not be completed for all axes specified (e.g.: cancelation due to: Servo Enable not provided, measuring system switchover, release of direction key, and the like).

Reaction

NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Check possibilities of canceling:

- Servo enable missing (V 380x0001.1)
- Traversing keys + or - missing (V 380x0004.6 and .7)
- Feed override = 0

Which axes are involved in channel-specific referencing is determined by the axis-specific MD 34 110 **REFP_CYCLE_NR.**

-1: **No** channel-specific referencing, NC start **without** referencing

0: **No** channel-specific referencing, NC start **with** referencing

1-8: Channel-specific referencing. The entered number corresponds to the referencing order. (When all axes with contents 1 have reached the reference point, the axes with contents 2 will start, etc.).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

20006 Channel %1 axis %2 reference point shutdown velocity not reached

Explanation

%1 = channel number
%2 = axis name, spindle number

During the 2nd phase of reference-point approach (waiting for zero mark), the end of the cam has been reached, but the reference-point approach speed was not in the tolerance window. (This is possible if the axis rests already at the end of the cam. Phase 1 is thus considered as already completed and will not be started).

The 2nd phase is aborted (in this case, before the cam) and the reference point approach is automatically restarted with phase 1. If the approach speed is also not reached with the 2nd attempt, the reference point approach is finally aborted and an alarm is displayed.

Approach speed:	MD 34 040 **REFP_VELO_SEARCH_MARKER**
Speed tolerance:	MD 35 150 **SPIND_DES_VELO_TOL**

Reaction

NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy

Please inform the authorized personnel/customer service.
MD for the approach speed MD 34 040 Reduce **REFP_VELO_ SEARCH_ MARKER** and/or increase the MD for the speed tolerance MD 35 150 **SPIND_DES_VELO_TOL**.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

20007 Channel %1 axis %2 Reference point approach requires 2 measuring systems

Explanation
%1 = channel number
%2 = axis name, spindle number

The setting MD 34200 **ENC_REFP_MODE** = 6 requires 2 encoders!

Reaction
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Either modify referencing mode MD **34200 ENC_REFP_MODE** or install and configure a second encoder.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

20008 Channel %1 axis %2 Reference point approach requires a second referenced measuring system

Explanation
%1 = channel number
%2 = axis name, spindle number

The setting MD 34200 **ENC_REFP_MODE** = 6 is not applicable to this control system.

Reaction
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy
Modify referencing mode via **MD 34200 ENC_REFP_MODE**.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

20050 Channel %1 axis %2 Handwheel traversing active

Explanation
%1 = channel number
%2 = axis name, spindle number

The axes cannot be traversed using the conventional method, since traversing is still being carried out via the handwheel.

Reaction
Alarm display.

Remedy
Decide whether you want to traverse the axis via the direction keys or via the handwheel. Exit handwheel traverse, if necessary with "Clear axis distance to go" (V 380x0002.2).

Program continuation by
The alarm display will disappear with the alarm cause. No further operation required.

20051 Channel %1 axis %2 Handwheel traversing not possible

Explanation
%1 = channel number
%2 = axis name, spindle number

The axis is already traversed via the direction keys; traversing via handwheel is thus no longer possible.

Reaction
Alarm display.

Remedy
Decide whether you want to traverse the axis via the direction keys or via the handwheel.

Program continuation by: The alarm display will disappear with the alarm cause. No further operation required.

20052

Channel %1 axis %2 already active

Explanation:

%1 = channel number
%2 = axis name, spindle number

The axis is to be traversed as a machine axis in the JOG mode using the direction keys on the machine control panel. This, however, is not possible because:

1. it is already traversed as a geometry axis (V32001000.6 V32001000.7, V32001004.6 V32001004.7 or V32001008.6 V32001008.7)
2. it is already traversed as a machine axis (V38000004.6 V38000004.7) or
3. a frame is valid for a rotated coordinate system and another geometry axis involved is already traversed in the JOG mode using the direction keys.

Reaction: Alarm display.

Remedy: Stop traversing via the channel or axis interface or stop the other geometry axis.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

20055

Channel %1 Master spindle does not exist in the JOG mode

Explanation:

%1=channel number

The displayed axis is to be traversed as a machine axis at revolutional feedrate in the JOG mode, but no master spindle is defined from which the actual speed could be derived.

Reaction: Alarm display.
Interface signals are set.

Remedy: Please inform the authorized personnel/customer service.
If the revolutional feedrate is also to be active in the JOG mode, a master spindle must be defined via the channel-specific machine data 20090 **SPIND_DEF_MASTER_SPIND**. Then, in the **PARAMETERS** operating area, use the softkeys "SETTING DATA" and "JOG DATA" to open a screen in which you will select the G function **G95**. Then you can enter the JOG feedrate in [mm/rev]. (If 0 mm/rev. is selected for the JOG feedrate, the control system will use the value from the axis-specific MD 32050 **JOG_REV_VELO** or - in the case of rapid traverse override - 32040 **JOG_ REV_VELO_RAPID).**

The revolutional feedrate in the JOG mode is deactivated by using the **G94** function instead of the **G94** function.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

20056

Channel %1 axis %2 No revolutional feedrate since axis/spindle %3 stopped

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = axis name, spindle number

An axis is to be traversed at revolutional feedrate in the JOG mode, but the feedrate of the spindle/axis from which the feedrate is to be derived is 0.

Reaction

Alarm display.

Remedy

Traverse the spindle/axis from which the feedrate is to be derived.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

20057

Channel %1 block %2 Revolutional feedrate for axis/spindle %3 <= zero.

Explanation

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

Revolutional feedrate was programmed for an axis/spindle, but no velocity was programmed or the programmed value is less than zero.

Reaction

Alarm display
Interface signals are set
NC Stop at alarm
NC not ready
NC Start inhibited
Channel processing not ready

Remedy

Please inform the authorized personnel/customer service.
Correct the part program.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

20058

Channel %1 axis %2 Revolutional feedrate:Illegal feedrate source

Explanation

%1 = channel number
%2 = axis name, spindle number

An axis/spindle is to be traversed at revolutional feedrate. The reference axis/spindle defined in SD 43300 **ASSIGN_FEED_PER_REV_SOURCE** points to itself. The resulting coupling cannot be executed.

Reaction

Alarm display.

Remedy

The spindle/axis from which the feedrate is to be derived points to itself.

Program continuation by

The alarm display will disappear with the alarm cause. No further operation required.

20060

Channel %1 axis %2 cannot be traversed as a geometry axis

Explanation

%1 = channel number
%2 = axis name:

The axis is currently not in the "Geometry axis" status". It can therefore not be traversed as a geometry axis in the JOG mode.

If the abbreviation **WCS** (workpiece coordinate system) appears in the Position display window, only the **geometry axes** can be traversed using the direction keys. (**MCS** ... machine coordinate system; using the direction keys of the machine control panel, **all machine axes** can now be traversed).

Reaction: Alarm display.

Remedy: Check the operation to make sure whether really geometry axes are to be traversed; otherwise, use the **WCS/MCS** key on the machine control panel to switch over to the machine axes.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

20062

Channel %1 axis %2 already active

Explanation: %1 = channel number
%2 = axis name, spindle number

The displayed axis already traverses as a machine axis. It can therefore not be run as a geometry axis.

In the JOG mode, an axis can be traversed using two different interfaces.

1. **as a geometry axis**via the channel-specific interface
   1. GEO axis: V32001000.6 and .7
   2. GEO axis: V32001000.6 and .7
   3. GEO axis: V32001008.6 and .7
2. as a **machine axis:**via the axis-specific interface
   DB 31 - DB 48 DBX8.6 or DBX8.7

The standard machine control panel cannot be used to run a machine axis both as a machine and geometry axis.

Reaction: Alarm display.

Remedy: Start the geometry axis only when the traversing motion as a machine axis is completed.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

20065

Channel %1 Master spindle not defined for geometry axes in the JOG mode

Explanation: %1=channel number

The displayed axis is to be traversed as a geometry axis at revolutional feedrate in the JOG mode, but no master spindle is defined from which the actual speed could be derived.

Reaction: Alarm display.
Interface signals are set.

Remedy: If the revolutional feedrate is also to be active in the JOG mode, a master spindle must be defined via the channel-specific machine data 20090 **SPIND_DEF_MASTER_SPIND**. Then, in the **PARAMETERS** operating area, use the softkeys "SETTING DATA" and "JOG DATA" to open a screen in which you will select the G function **G95**. Then you can enter the JOG feedrate in [mm/rev]. (If 0 mm/rev. is selected for the JOG feedrate, the control system will use the value from the axis-specific MD 32050 **JOG_REV_VELO** or - in the case of rapid traverse override - 32040 **JOG_ REV_VELO_RAPID).**

The revolutional feedrate in the JOG mode is deactivated by using the **G94** function instead of the **G94** function.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

20090

Axis %1 Traversing to fixed stop not possible. Check programming and axis data.

Explanation

%1 = axis name, spindle number

1. The function "Travel to fixed stop" was programmed with FXS[AX]=1, but this function is not (yet) supported by the axis. Check MD 37000 FIXED_STOP_MODE. This function is not available for simulated axes.
2. No motion was programmed for the AX axis when selecting it. AX is a machine axis identifier.
3. A traversing motion must always be programmed in the selection block for the axis/spindle for which the function is activated.

Reaction

Mode group not ready.
Under certain circumstances, it can be switched across all channels via MD.
Channel not ready.
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy

Please inform the authorized personnel/customer service.

- Check axis type
- Check MD 37000
- Is the programming of a motion of the machine axis missing in the selection block?

Program continuation by

Use the RESET key to cancel the alarm.

20091

Axis %1 has not reached the fixed stop

Explanation

%1 = axis name, spindle number

When trying to approach a fixed stop, either the programmed end position was reached or the traversing motion was canceled.
This alarm can be skipped via the machine data $MA_FIXED_STOP_ALARM_MASK.

Reaction

Mode group not ready.
Under certain circumstances, it can be switched across all channels via MD.
Channel not ready.
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy

Correct the part program and its settings accordingly:

- Was the traversing block canceled?
- If the axis position is to correspond to the programmed end position, then correct the end position.
- If the programmed end position is located in the part, check the triggering criterion.
- Was the contour deviation resulting in the triggering overdimensioned? Is the torque limit set too high?

Program continuation by

Use the RESET key to cancel the alarm.

20092

Axis %1 Travel to fixed stop still active

Explanation
%1 = axis name, spindle number

It was tried to move the axis traveled to fixed stop while it stands at the stop or when the selection is not yet completed.

Reaction
Under certain circumstances, it can be switched across all channels via MD.
Channel not ready.
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Check the following:

- Is the axis at the fixed stop also moved due to a traversing motion of geometry axes?
- Is a deselection carried out although the axis stands at the fixed stop?
- Was the selection interrupted via RESET?
- Has the PLC switched the acknowledgment signals?

Program continuation by
Use the RESET key to cancel the alarm.

20093

Axis %1 Zero speed monitoring at stop has tripped

Explanation
%1 = axis name, spindle number

The position of the axis since the selection has been carried out is across the zero speed window.

Reaction
Under certain circumstances, it can be switched across all channels via MD.
Channel not ready.
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.

- Check the mechanics, e.g. stop broken?
  Has the part to be clamped given?
- Position window for the zero speed monitoring too small
  (37020 MD: $MA_FIXED_STOP_WINDOW_DEF)
  (43520 Setting Data: $SA_FIXED_STOP_WINDOW).
  Default value 1 mm each.

Program continuation by
Use the RESET key to cancel the alarm.

20094

Axis %1 Function was canceled

Explanation
%1 = axis name, spindle number

the function was canceled.
Possible reasons are:

- As a result of a pulse inhibitor, the torque can no longer be established.
- The PLC has reset the acknowledgments.

The alarm can be reconfigured using MD
ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction
Mode group not ready.
Under certain circumstances, it can be switched across all channels via MD.

Channel not ready.
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Is there a pulse inhibit provided from the infeed/regenerative feedback unit or from the LPC? Have the acknowledgment bits been deleted by the PLC even if no deselection was requested by the NCK?

Program continuation by
The RESET key must be used in all channels of this mode group to cancel this alarm.

21610 Channel %1 axis %2 %3 frequency exceeded

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = string (encoder number)

The admissible maximum frequency of the active encoder specified in the axis-specific machine data 36 300 **ENC_FREQ_LIMIT [n]** (n ... encoder number, 1 or 2) (axis-specific interface signal V390x0000.2) **has been exceeded. The reference of the actual value to the mechanical carriage position could have been lost.**

The alarm can be reconfigured using MD 11412 **ALARM_REACTION_CHAN_NOREADY** (NC not ready).

Reaction
Alarm display.
Interface signals are set.
NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Stop at alarm.
NC Start inhibited.

Remedy
Check MD 36300 **ENC_FREQ_LIMIT [0]** .

Program continuation by
The RESET key must be used in all channels of this mode group to cancel this alarm.

21612 Channel %1 axis %2 VDI signal 'Servo enable' reset during the traversing motion

Explanation
%1 = channel number
%2 = axis name, spindle number

The Servo Enable interface signal (V 380x0002.1) has been set to zero for the displayed axis, although an axis of the geometry group has been moved.

As axes of the geometry compound, the axes entered in the MD array 20050 **AXCONF_GEOAX_ASSIGN_TAB** are considered. Servo Enable must be provided for **all** existing geometry axes, irrespective whether they are currently moving or not.

Reaction
NC Start inhibited.
Alarm display.
Interface signals are set.
NC Stop at alarm.

Remedy
Check the interface signal "Servo enable" (V 380x0002.1, e.g. using the PLC status display in the DIAGNOSIS operating area). Trace the signal back to the sections in the PLC user program from which the signal is linked and set/deleted.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

21614 Channel %1 axis %2 hardware limit switch %3

Explanation

%1 = channel number
%2 = axis name, spindle number
%3 = string (+, - or +/-)

The VDI signal "Hardware limit switch" (V 380x1000.0 or 1) has been set at the NC/PLC interface.

Reaction

Alarm display.
NC Start inhibited.

Remedy

1. With axes already approached to the reference point, software switch 1 or 2 should respond before the hardware limit switch is reached.
   Check **POS_LIMIT_PLUS, POS_LIMIT_MINUS, POS_LIMIT_ PLUS2** and **POS_LIMIT_MINUS2** (MD 36 100 - 36 130), as well as the interface signal for the selection 1st/2nd software limit switch (V 380x1000.2 and .3) and correct if necessary (PLC user program).
2. If the axis has not yet been approached to the reference point, you can leave the hardware limit switch in the JOG mode in the opposite direction.
3. Check the PLC user program and the link from the switch to the PLC input unit if the axis has not at all reached the hardware limit switch.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

21617 Channel %1 block %2 Transformation does not permit passing the pole

Explanation

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

The specified course of the characteristic passes the pole or a forbidden area of the transformation.

Reaction

Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy

Modify the part program (if the alarm has occurred in the AUTO mode).

To retract from the alarm position, deselect the transformation (RESET alone is not sufficient if the transformation remains active even after RESET).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

21619 Channel %1 block %2 Transformation active: Motion not possible

Explanation

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

The specified motion is not possible due to the machine kinematics. Transformation-depending error causes may be:
TRANSMIT:
A (circular) area exists around the pole to which positioning is not possible. This area is due to the fact that the tool reference point cannot be traversed into the pole. The area is defined by:

- the machine data ($MC_TRANSMIT_BASE_TOOL..)
- the active tool length compensation (see $TC_DP..).
  How the tool length compensation is taken into account depends on the work plane selected (see G17,..).

The machine stops at the margin of the area to which no positioning is possible.

Reaction
Alarm display
Interface signals are set
NC Stop at alarm
NC Start inhibited

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

Program continuation by
Please note: RESET alone is not sufficient even if the transformation remains also active after RESET.

Use the RESET key to cancel the alarm. Restart the part program.

21700

Channel %1 block %3 axis %2 Probe already deflected, signal transition not possible

Explanation
%1 = channel number
%2 =axis name, spindle number
%3 = block number

The probe programmed with the vocabulary word MEAS or MEAW is already deflected and has switched. For another measuring process, the probe signal must be reset (rest position of probe).

The axis display is not yet relevant; an axis-specific evaluation is planned for later development stages.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy
Check the starting position of the measuring process and/or check the sensing probe signals. Cable and plug o.k.?

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

21701

Channel %1 block %3 axis %2 Measuring not possible

Explanation
%1 = channel number
%2 =axis name, spindle number
%3 = block number

Measuring stage 2 (MEASA, MEAWA, MEAC)

The programmed measuring job is faulty.

Possible causes:

- illegal measuring mode
- invalid sensing probe
- invalid encoder
- invalid number of measuring edges
- the same measuring edges can only be programmed in the mode 2
- invalid FIFO number
- The number of programmed FIFOs does not correspond to the number of sensing probes used in the measuring job.

Further causes

- A measuring job is already active (e.g. from a synchronized action).

Reaction: Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy: Correct the measuring jobs.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

21702

Channel %1 block %3 axis %2 Measuring was canceled

Explanation: %1 = channel number
%2 =axis name, spindle number
%3 = block number

The measuring block is completed (the programmed end position of the axis was reached), but the activated sensor has not yet responded.

Reaction: Alarm display.

Remedy: Check the traversing motion in the measuring block.

- Must the activated sensor have in any case been switched up to the specified axis position?
- Sensor, cable(s), cable distributor, terminal connections o.k.?

Either program all GEO axes explicitly or program the traversing movement using the POS[axis] command.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

21703

Channel %1 block %3 axis %2 Probe not yet deflected, illegal signal transition

Explanation: %1 = channel number
%2 =axis name, spindle number
%3 = block number

The selected probe is not (!) deflected and can therefore not acquire a measured value from the deflected to the non-deflected state.

Reaction: Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy: - Check probe
- Check starting position for measuring
- Check program

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

21800

Channel %1 Number of required workpieces = %2 reached

Explanation: %1 = channel number
%2 = required workpieces

This alarm is activated via the MD 27880 **PART_COUNTER**, bit 1: The number of workpiece counted {$AC_ACTUAL_PARTS or $AC_SPECIAL_PARTS} is equal to or already greater than the programmed value for the number of required workpieces {$AC_REQUIRED_PARTS}. At the same time, the channel VDI signal "Required number of workpieces reached" is output. The value of the workpieces counted $AC_ACTUAL_PARTS is set to zero whereas the value of $AC_SPECIAL_PARTS is kept.

Reaction	Alarm display Interface signals are set NC not ready
Remedy	No program interruption. Clear alarm display.
Program continuation by	Press the "Cancel" key to cancel the alarm.

22000

Channel %1 block %3 spindle %2 No gear stage change possible

Explanation

%1 = channel number
%2 = spindle number
%3 = block number, label

Automatic gear change selection with M40 has been programmed. The new M word is not in the currently selected gear stage, but the spindle is not in **"control mode".**

The automatic gear stage change (M40 in conjunction with the spindle speeds under address S) requires **"control mode"** of the spindle.

Reaction

Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy

Change to control mode of spindle in front of the S word that requires gear stage change.

The following is used to switch to the control mode:

- **M03, M04, M05** or **M41 .. M45** from axis mode and positioning mode
- Interface signal "Gear switched" (V 38032000.3) from the oscillation mode

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

22010

Channel %1 block %3 spindle %2 Actual gear stage does not correspond to requested gear stage.

Explanation

%1 = channel number
%2 = spindle number
%3 = block number, label

The required gear stage change has been completed.
The actual gear stage signaled (activated) from the PLC does not match with the nominal gear stage requested from the NC.
Note: It is recommended to use always the requested gear stage.

Reaction

Alarm display.

Remedy

Correct the PLC program.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

22011 Channel %1 block %3 spindle %2 Switching to programmed gear stage not possible

Explanation
%1 = channel number
%2 = spindle number
%3 = block number, label

When deselecting the DryRun and Program Test functions, it is not possible to switch a gear stage change in the REPOS module back to a gear stage earlier programmed. This is the case if the spindle in the deselection block is active not in the speed-controlled mode, but as a following axis or within a transformation. Switching back of a gear stage change when deselecting the function mentioned above is avoided by resetting bit2 of the machine data 35035 SPIND_FUNCTION_MASK.

Reaction
Alarm display.

Remedy
Change the deselection block or the block search target block to speed-controlled mode (M3, M4, M5, SBCOF). .

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

22020 Channel %1 block %3 spindle %2 Gear stage change position not reached

Explanation
%1 = channel number
%2 = spindle number
%3 = block number, label

By configuring MA_GEAR_STEP_CHANGE_ENABLE[AXn] = 2, the spindle is traversed to the position stored in MA_GEAR_STEP_CHANGE_POSITION[AXn] prior to the actual gear stage change. The requested gear stage change position was not reached.

Reaction
Channel not ready.
NC Start inhibited in this channel.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Correct the sequence in the PLC.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22050 Channel %1 block %3 spindle %2 No transition from speed control to position control mode

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = block number, label

- Oriented spindle stop (SPOS) was programmed, but no spindle encoder is defined.
- When position control is switched on, the spindle speed is greater than the limit speed of the measuring system.

Reaction
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Spindle without installed encoder: Do not use NC language elements that require encoder signals.

Spindle with installed encoder: Enter spindle encoder via MD 30200 **NUM_ENCS**.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22051

Channel %1 block %3 spindle %2 Reference mark not found

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = block number, label

When determining the reference point, the spindle rotated a larger path than defined in the axis-specific MD 34060 **REFP_MAX_MARKER_DIST**, without receiving a reference mark signal. The check is carried out when the spindle is positioned with SPOS or SPOSA if the spindle has first not been run with speed control (S=...).

Reaction
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Check machine data 34060 **REFP_MAX_MARKER_DIST**and correct it if necessary. The entered value specifies the distance to be traversed between two zero reference marks in [mm] or [degrees].

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22052

Channel %1 block %2 axis %3 No standstill at block change

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = block number, label

The spindle displayed was programmed either as a spindle or as an axis although a positioning process arising from the previous block is still running (with SPOSA ... Position spindle across block limits).

Example:N100 SPOSA [2] = 100
:
N125 S2 = 1000 M2 = 04 ; error if spindle S2 programmed in
; block N100 is still running!

Reaction
NC Start inhibited.
NC Stop at alarm.
Alarm display.
Interface signals are set.

Remedy
Before reprogramming the spindle/axis after the SPOSA statement, waiting for the programmed spindle position should be triggered via a WAITS command.

Example:N100 SPOSA [2] = 100
:
N125 WAITS (2)
N126 S2 = 1000 M2 = 04

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22053

Channel %1 block %3 spindle %2 Referencing mode is not supported

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = block number, label

With SPOS and using an absolute encoder, only referencing mode MD **34200 ENC_REFP_MODE** = 2 is supported! **ENC_REFP_MODE** = 6 is generally not supported by SPOS!

Reaction	NC Start inhibited. NC Stop at alarm. Alarm display. Interface signals are set.
Remedy	Change setting of MD 34200 **ENC_REFP**_MODE, switch to JOG + REF and then approach the reference point.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

22055 Channel %1 block %3 spindle %2 Configured positioning speed too large

Explanation	%1 = channel number %2 = axis name, spindle number %3 = block number, label The current position is not referenced with the encoder position although reference is made to it.
Reaction	Alarm display.
Remedy	Correct the part program. Establish the zero mark position either by positioning, rotation (at least 1 revolution) in the speed-controlled mode or via G74 before enabling the function generating the alarm.
Program continuation by	Use the "Cancel" key to cancel the alarm. No further operation required.

22062 Channel %1 axis %2 Reference point approach:Zero mark searching speed (MD) is not reached

Explanation	%1 = channel number %2 = axis name, spindle number The configured zero mark searching speed is not reached.
Reaction	Alarm display. Interface signals are set. NC Start inhibited NC Stop at alarm
Remedy	Check the active speed limitation. Configure the lower zero mark searching speed in MD 34040 **REFP_VELO_SEARCH_MARKER**. Check the tolerance range for the actual speed MD 35150 **SPIND_DES_VELO_TOL**. Set a different referencing mode MD 34200 **ENC_REFP_MODE=7.**
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

22064 Channel %1 axis %2 Reference point approach:Zero mark searching speed (MD) too high

Explanation	%1 = channel number %2 = axis name, spindle number The configured zero mark searching speed is too high. The encoder frequency for the active measuring system is exceeded.
Reaction	Alarm display. Interface signals are set. NC Start inhibited NC Stop at alarm
Remedy	Configure the lower zero mark searching speed in MD 34040 **REFP_VELO_SEARCH_MARKER**. Check encoder limit frequency configuration MD 36300 **ENC_FREQ_LIMIT** and MD 36302 **ENC_FREQ_LIMIT_LOW**.

Set a different referencing mode MD 34200 **ENC_REFP_MODE=7.**

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

22067 Channel %1 tool management: Tool change not possible, since no tool ready for use is in tool group %2

Explanation: %1 = channel ID
%2 = string (identifier)

The desired tool change is not possible. The specified tool group does not possess a replacement tool ready for use which could be loaded. Possibly, all tools which could be used have been set to the "Disabled" status by the tool monitoring.

Reaction: Interface signals are set.
Alarm display.
NC Start inhibited.
NC Stop at alarm at block end.
Stop at alarm .

Remedy:
- Make sure that a tool ready for use is contained in the specified tool group at the time when the requesting tool change occurs.
- This can be achieved, e.g. by replacement of disabled tools
- or also by enabling a tool manually.
- Check whether the tool data are defined correctly.
  Have all tools of the group which are intended for work been defined using the specified identifier? Have they all been loaded?

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

22100 Channel %1 block %3 spindle %2 Chuck speed exceeded

Explanation: %1 = channel number
%2 = axis name, spindle number
%3 = block number, label

The actual speed of the displayed spindle is greater than defined in the axis-specific MD **35100 SPIND_VELO**_LIMIT plus the tolerance set in MD **35150 SPIND_DES_VELO**_TOL.

If the drive manufacturer has carried out the optimization correctly, this alarm cannot occur!

Reaction: Alarm display.
Interface signals are set.
NC not ready.
NC Stop at alarm.
NC Start inhibited.

Remedy: Check start-up and optimization data against the Start-Up Guide of the drive manufacturer and correct them.

Increase the tolerance window in MD **35150 SPIND_DES_VELO_TOL**.

Program continuation by: Use the RESET key to cancel the alarm.

22101 Channel %1 block %3 spindle %2 Maximum speed for actual-value coupling exceeded

Explanation
%1 = channel number
%2 = axis name, spindle number
%3 = block number, label

The limit frequency of the active encoder (axis-specific MD 36300 **ENC_FREQ_LIMIT[0]) has been exceeded for one of the functions G33 (thread cutting with encoder), G95 (revolution feedrate)** or **G96 (constant cutting rate)**. The spindle synchronization got lost as a result of this.

Reaction
Alarm display.
Interface signals are set.
NC Stop at alarm.
NC Start inhibited.

Remedy
Check whether the encoder has been enabled by the interface signal: **position measuring system** V380x0001.5 or whether the limit frequency for this encoder has been loaded with the correct default value in MD 36300 **ENC_FREQ_LIMIT[0]**.

Check the maximum spindle speed in the axis-specific MD 35130 **GEAR_STEP_MAX_VELO_LIMIT** and correct (reduce) it if necessary.

Program an upper spindle speed limitation with **G26 S..** in the previous NC block which must be below the maximum encoder limit frequency.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22200 Channel %1 block %3 spindle %2 Axis stop when tapping

Explanation
%1 = channel number
%2 = axis name, spindle number

The drilling axis has been stopped via the NC/PLC interface during tapping with compensation chuck (G63) - the spindle continues rotating. As a result, the thread and possibly also the tap have been damaged.

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy
Provide for an interlock in NC user program so that no axis stop can be initiated when tapping is active. If the tapping operation is to be aborted in critical machine conditions, spindle and axis must be stopped at the same time if possible. Slight deviations will be compensated by the compensation chuck.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22250 Channel %1 spindle %2 Axis stopped in thread cutting

Explanation
%1 = channel number
%2 = axis name, spindle number

The thread cutting axis has been stopped during an active thread block.

The stop can be caused by VDI signals resulting in feed interruption.

Reaction
Alarm display.
Interface signals are set.
NC Start inhibited.

Remedy
Check the axis/spindle-specific stop signals (V 380x0004.3).

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

22260

Channel %1 spindle %2 Thread might be damaged

Explanation

%1 = channel number
%2 = axis name
%3 = block number

When **DECODING SINGLE BLOCK** is selected and several thread blocks are linked with each other, machining breaks lasting as long as the next following block is executed with NC restart arise at the end of blocks.

When the control system is operated in normal single block mode, a higher-level logic stops the program only at the end of those blocks at which no contour falsifications or contour errors might occur. In the case of chained thread blocks, this is after the last thread block!

Reaction

Alarm display.

Remedy

If only one thread block is programmed, the alarm message can be ignored.

In the case of several thread blocks following one after another, do not execute this machining step in the automatic mode **DECODING SINGLE BLOCK**.

Program continuation by

Press NC START to cancel the alarm and continue the program execution.

22270

Channel %1 block %2 spindle %3 Spindle speed too high for thread cutting

Explanation

%1 = channel number
%2 = block number, label
%3 = axis name, spindle number

The spindle speed for thread cutting G33 is so high that the maximum axis velocity is exceeded due to the programmed thread pitch.

Reaction

Alarm display.

Remedy

Program either a lower spindle speed or a speed limitation via G26 S ... or reduce the spindle speed prior to the thread block via SD 43 220 **SPIND_MAX_VELO_G26** or the spindle override.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

22275

Channel %1 block %2 Zero velocity of velocity axis reached at position %3

Explanation

%1 = channel number
%2 = block number, label
%3 = position

When thread cutting using G35 axis standstill was reached at the specified position, caused by the linear reduction of the thread pitch.
The standstill position of the thread axis depends on:
- the programmed pitch reduction
- the thread length.

Reaction

Alarm display.

Remedy

Change at least one of the a/m influencing factors.

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

22280

Channel %1 in block %2: Progr. acceleration-up travel too short %3; %4 required

Explanation

%1 = channel number
%2 = block number, label
%3 = prog. acceleration travel

%4 = required acceleration travel

To be able to observe the programmed acceleration travel, the thread axis was overloaded with regard to the acceleration. To be able to accelerate the axis with the configured dynamics, the length of the acceleration travel must be at least as large as specified in the parameter %4.

Reaction: Alarm display.

Remedy: Adapt the SD **42010 THREAD_RAMP_DISP** accordingly.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

25000

Axis %1 Hardware error active encoder

Explanation: %1 = axis name, spindle number

The signals of the position actual encoder (interface signal V 380x001.5) currently active are missing, are not in phase or have a short-circuit/short circuit to ground.

Reaction:
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switched to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference point).

Remedy: Check the contacts of the **measuring circuit connector**. Check the encoder signals; in case of errors, change the encoder.

Program continuation by: Power ON

25010

Axis %1 Measuring system contaminated

Explanation: %1 = axis name, spindle number

The encoder used for the position control provides a contamination signal (only with measuring systems generating a contamination signal).

Reaction:
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference point).

Remedy: Check the measuring system with respect of the settings of the measuring system manufacturer.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

25011

Axis %1 Measuring system contaminated

Explanation: %1 = axis name, spindle number

The encoder not used for the position control provides a contamination signal (only with measuring systems generating a contamination signal).

Reaction: Alarm display.

Remedy
Please inform the authorized personnel/customer service.
Check the measuring system with respect of the settings of the measuring system manufacturer.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

25020

Axis %1 Zero mark monitoring

Explanation
%1 = axis name, spindle number

The pulses of the position encoder between 2 zero mark pulses are counted (hardware function). It is checked whether the encoder always provides the same number of pulses between the zero marks. As soon as a deviation in the 4 less-significant bits of the counter is ignored, an alarm is output.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference point).

Remedy
The deviations might have been caused by transmission errors, interference, encoder hardware errors or errors in the evaluation electronics of the encoder used for position control. Therefore, the following should be checked in the actual value branch:

1. **Transmission path:** Check the actual value connector on the motor for correct contacting; check encoder cable for continuity, short circuit and short circuit to ground (loose contact?).
2. **Encoder pulses:** Encoder supply within tolerance limits?
3. **Evaluation electronics:** Change/reconfigure the drive module used. The monitoring can be switched off by setting MD 36310 310 **ENC_ZERO_MONITORING** to "0".

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

25021

Axis %1 Zero mark monitoring

Explanation
%1 = axis name, spindle number

The monitoring refers to the encoder not used by the position control! (IS DB 31 - 48, DBX 1.5 = 0 or 1.6 = 0)

The pulses of the position encoder between 2 zero mark pulses are counted (hardware function). It is checked in the interpolation clock matrix (default setting
4 ms) whether the encoder always provides the same number of pulses between the zero marks. If a deviation is registered in the 4 less-significant bits of the counter, an alarm is output.

Reaction
Alarm display.

Remedy
Please inform the authorized personnel/customer service.
The deviations might have been caused by transmission errors, interference, encoder hardware errors or errors in the evaluation electronics of the encoder used for position control. Therefore, the following should be checked in the actual value branch:

1. **Transmission path:**Check the actual value connector on the motor and on the FDD module for correct contacting; check encoder cable for continuity, short circuit and ground connection (loose contact?).

2. **Encoder pulses:** Encoder supply within tolerance limits?
3. **Evaluation electronics:**Change/reconfigure the drive module used.

The monitoring can be deactivated by setting the machine data **ENC_ZERO_MON_ACTIVE [n]=...** (n ... encoder number: 1, 2) to 0.

Program continuation by: Use the "Cancel" key to cancel the alarm. No further operation required.

25030

Axis %1 Actual velocity alarm limit

Explanation: %1 = axis name, spindle number

The actual velocity of the axis is checked cyclically in IPO clock. If there is no error, the actual speed can never be higher than defined in the axis-specific MD 36200 200 **AX_VELO_LIMIT** (threshold value for speed monitoring). This threshold value specified in [mm/min, rev./min] must be entered by approx. 5 ... 10 % greater than it may occur with the maximum traversing rate. Drive errors might result in exceeding the speed what causes the alarm.

Reaction:
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy:
Check speed setpoint cable (bus cable).
Check actual values and position-control direction.
Change position-control direction if the axis rotates uncontrolled axis-specific MD 32110 110 **ENC_FEEDBACK_POL** = < -1, 0, 1.→
Increase monitoring limit value in MD 36200 200 **AX_VELO_LIMIT**.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

25040

Axis %1 Zero speed control

Explanation: %1 = axis name, spindle number

The NC monitors holding of the position at standstill. The monitoring is started after an axis-specific time set in MD 36040 040 **STANDSTILL_DELAY_TIME** after the interpolation has been completed. It is checked continuously whether the axis remains within its tolerance threshold set in MD 36030 030 **STANDSTILL_POS_TOL**.

The following cases are possible:

1. The interface signal SERVO ENABLE (V 380x0002.1) is zero, as the axis is clamped mechanically. Mechanic influences (e.g. high forces due to machining) cause the axis to leave its permissible position tolerance.
2. With the position control circuit closed (without clamping) - interface signal SERVO ENABLE (V 380x0002.1) IS "1" - high mechanical forces with low gain in the open-loop position control circuit cause the axis to leave its position.

Reaction:
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy:
- Check MD 36040 **STANDSTILL_DELAY_TIME** and MD 36030 **STANDSTILL_POS_TOL** and increase it as necessary.

- Estimate the **machining** force and - if necessary - reduce them by reducing the feedrate/increasing the speed.
- Increase the **clamping pressure**.
- Increase the **gain** in the position control loop by enhancing the optimization (loop gain factor MD 32200 **POSCTRL_GAIN**).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

25050

Axis %1 Contour monitoring

Explanation

%1 = axis name, spindle number

The NC calculates the actual value for each interpolation intermediate point (setpoint) of an axis, which should result from an internal model. If the difference amount between this calculated actual value and the real machine actual value is greater than the amount set in MD **CONTOUR_TOL**, the program is aborted and the alarm output.36400

Reaction

NC not ready
Under certain circumstances, it can be switched for several channels via MD.
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy

- **Check the tolerance value in** MD 36400 **CONTOUR_TOL** to see whether the programmed value is too small.
- **Check the optimization of the position controller (loop-gain factor** in MD 32 200 **POSCTRL_GAIN**) to make sure that the axis follows the setpoint setting without exceeding the amplitude. Otherwise, speed controller optimization must be enhanced or the loop-gain factor (K_V factor) be reduced.
- **Check the acceleration in** MD 32 300 **MAX_AX_ACCEL**. If the current limits are reached because of too high acceleration, the open-loop position control is disconnected as a result. The "lost" actual value is recovered in the form of an overshoot as soon as the open-loop position control circuit is closed again.
- Improve the **speed controller optimization.**
- Check the **mechanics (smooth running, rotational masses)**.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

25060

Axis %1 Speed setpoint limitation

Explanation

%1 = axis name, spindle number

The speed setpoint has exceeded its upper limit longer than allowed.

The maximum set speed is limited to a certain percentage value using the axis-specific MD 36210 **CTRLOUT_LIMIT**. The input value of 100% corresponds to the rated speed of the motor and thus the rapid traverse rate.

Short-time overshooting is tolerated provided they do not last longer than admitted by the axis-specific MD 36220 **CTRLOUT_LIMIT_TIME**. During this time, the set value is limited to the set maximum value (MD 36210 **CTRLOUT_LIMIT**).

Reaction

NC not ready
Under certain circumstances, it can be switched for several channels via MD.
NC Start inhibited.

NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy

If the drive controller is set correctly and normal machining conditions are provided, this alarm should not occur.

- **Check the actual values:**local heavy running of the carriage, speed dip and torque peak in case of workpiece/tool contact, traversing to a fixed obstacle, etc.
- **Check the direction of the position control:** Does the axis move uncontrolled?
- Check the speed setpoint cable.

Program continuation by

Use the RESET key to cancel the alarm.

25070

Axis %1 Drift limit too large

Explanation

%1 = axis name, spindle number

Only with analog drives.

The permissible maximum value of the drift (internal drift value integrated to the automatic drift compensation) has been exceeded during compensation! The permissible maximum value is defined in the axis-specific MD 36710 710 **DRIFT_LIMIT**. The drift value itself is not limited.

Automatic drift compensation: MD 36700 700 **DRIFT_ENABLE** = 1
The deviation of the actual position to the set position (drift) is automatically checked cyclically in IPO clock and automatically compensated to zero by slowly integrating an internal drift value.

Manual drift compensation: MD 36700 700 **DRIFT_ENABLE** = 0
A static offset can be added to the speed setpoint in MD 36 720 **DRIFT_VALUE**. It will not be included in the speed monitoring, since it acts as a voltage zero offset.

Reaction

Alarm display.

Remedy

Readjust drift compensation with automatic drift compensation on the drive switched off until the following error is approximately zero. Then reactivate automatic drift compensation to compensate dynamic drift changes (heating).

Program continuation by

Use the "Cancel" key to cancel the alarm. No further operation required.

25080

Axis %1 Positioning monitoring

Explanation

%1 = axis name, spindle number

The axis of blocks in which "exact stop" is active must have reached the exact positioning window after the positioning time set in the axis-specific MD 36 020 **POSITIONING_TIME**.
Exact stop coarse: MD 36000 000 **STOP_LIMIT_COARSE**
Exact stop fine: MD 36010 010 **STOP_LIMIT_FINE**

Reaction

NC not ready
Under certain circumstances, it can be switched for several channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switched to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Check whether the exact stop limits (coarse and fine) match with the dynamic possibilities of the axes; otherwise, increase them - possibly in conjunction with the positioning time defined in MD 36020 **POSITIONING_TIME**.
Check the optimization of the speed controller/position controller; select the gain as high as possible.
Check the setting of the loop-gain factor (K_V factor) (MD 32200 **POSCTRL_GAIN**); increase if necessary.

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

25110 Axis %1 Selected encoder does not exist

Explanation
%1 = axis name, spindle number

The selected encoder does not match with the maximum number of encoders specified in the axis-specific MD 30200 **NUM_ENCS**, i.e. the 2nd encoder does not exist.

Reaction
Alarm display.

Remedy
Please inform the authorized personnel/customer service.
Enter the number of actual-value sensors for this axis in machine data 30200 **NUM_ENCS** ("number of encoders"),

Input value 0: Axis without encoder **###** e.g. spindle

Input value 1: Axis with 1 encoder **###** default setting

Input value 2: Axis with 2 encoders **###** e.g. direct and indirect measuring systems

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

25200 Axis %1 Requested parameter set not permissible

Explanation
%1 = axis name, spindle number

For the position control, a new parameter set has been requested, the number of which is beyond the permissible limit (8 parameter sets: 0 ... 7 available).

Reaction
NC Stop at alarm.
NC Start inhibited.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Check the axis/spindle-specific interface signals (V380x4001.0 - .2 "Select parameter records A, B, C").

A parameter set includes the machine data:

- MD 31050 **DRIVE_AX_RATIO_DENOM [n]**
- MD 31060 **DRIVE_AX_RATIO_NUMERA [n]**
- MD 32200 **POSCTRL_GAIN [n]**
- MD 32810 **EQUIV_SPEEDCTRL_TIME [n]**
- MD 32910 **DYN_MATCH_TIME [n]**
- MD 36200 **AX_VELO_LIMIT [n]**

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

25201 Axis %1 Drive fault

Explanation
%1 = axis name, spindle number

The drive signals a fatal error of status class 1 (ZK1). The exact error cause can be derived from evaluating the additionally present drive alarms:

Alarm 300 500, alarms 300 502 - 300 505, alarm 300 508, alarm 300 515, alarm 300 608, alarm 300 612, alarm 300 614, alarms 300 701 - 300 761, alarm 300 799.

Reaction
NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Evaluate the above mentioned drive alarms.

Program continuation by
Use the RESET key to cancel the alarm.

25202 Axis %1 Waiting for drive

Explanation
%1 = axis name, spindle number

Drive group error (self-clearing)

Reaction
Alarm display.
Interface signals are set.

Remedy
We will wait for the drive.
This alarm reveals similar problems as alarm 25201 (see there).

It is present continuously during ramp-up when the drive does not communicate (e.g. Profibus connector disconnected).

Otherwise, the alarm is present only for a short time and is replaced in case of permanent problems with alarm 25201 after an internal time-out.

Program continuation by
Use the RESET key to cancel the alarm.

26000 Axis %1 Clamping monitoring

Explanation
%1 = axis name, spindle number

The clamped axis has been pushed out of its position. The permissible deviation is defined in the axis-specific MD 36050 050 **CLAMP_POS_TOL**.

Axis clamping is activated by the axis-specific interface signal
V 380x0002.3: "Clamping running". "Clamping process running" activated.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Determine the deviation of the actual position to the setpoint position and - depending on this - either increase the permissible tolerance in the MD or provide for mechanical improvement of the clamping (e.g. increase clamping pressure).

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

26001 Axis %1 Parameterization error:Friction compensation

Explanation

%1 = axis name, spindle number

It is not permitted to parameterize the adaptation characteristic in conjunction with quadrant error compensation, since the acceleration value 2 (MD 32560 **FRICT_COMP_ACCEL2** is not between acceleration value 1 (MD 32550 **FRICT_COMP_ACCEL1**) and acceleration value 3 (MD 32570 **FRICT_COMP_ACCEL3**).

Reaction

Mode group not ready.
Under certain circumstances, it can be switched across all channels via MD.
Channel not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy

Please inform the authorized personnel/customer service.
Check the setting parameters of the quadrant error compensation (friction compensation); if necessary disable the compensation via MD 32500 **FRICT_-COMP_ENABLE.**

Program continuation by

Use the RESET key to cancel the alarm.

26002 Axis %1 encoder %2 parameterization error:Increments

Explanation

%1 = axis name, spindle number
%2 = encoder number

1. Rotary measuring system (MD 31000 **ENC_IS_LINEAR**[] == FALSE)

   The number of increments set in MD 31020 **ENC_RESOL**[] does not match with the number of increments set in the drive machine data MD1005, or one of the two MDs is zero.

2. Absolute measuring system with EnDat interface (MD 30240 **ENC_TYPE**[] == 4)

   With absolute encoders, the resolution of the incremental and of the absolute track, which is provided from the drive, is additionally checked for consistency.

   - Motor measuring system: MD1005, MD1022
   - Direct measuring system: MD1007, MD1032

   The two drive machine data must be in a certain ratio one to another. Non-compliance with the below mentioned requirements results in an alarm.

2.1 Rotary measuring system (MD 31000 **ENC_IS_LINEAR**[] == FALSE)

MD1022/MD1005 == 4 * n [n=1,2,3...] (motor measuring system)

MD1032/MD1007 == 4 * n [n=1,2,3...] (direct measuring system)

2.2 Linear measuring system (MD 31000 **ENC_IS_LINEAR**[] == FALSE)

MD1005/MD1022 == 4 * n [n=1,2,3...] (motor measuring system)

MD10007/MD1032 == 4 * n [n=1,2,3...] (direct measuring system)

Reaction

NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Correct the machine data.
In the case of absolute encoders, drive alarms (if any) indicating encoder problems should be evaluated. These could be the cause for faulty entries in MD 1022/MD1032, which could be read from the encoder itself.

Program continuation by
Power ON

26003 Axis %1 Parameterization error:Leadscrew pitch

Explanation
%1 = axis name, spindle number

The pitch of the ballscrew/trapezoidal spindle set in the axis-specific MD 31030 **LEADSCREW_PITCH** is zero.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Determine the lead of the ballscrew (according to the machine manufacturer's specifications or measure the pitch with the spindle cover removed) and enter the value in MD 31030 **LEADSCREW_PITCH** (in most cases, 10 or 5 mm/rev).

Program continuation by
Power ON

26004 Axis %1 encoder %2 parameterization error:Increment spacing with linear encoders

Explanation
%1 = axis name, spindle number
%2 = encoder number

The scale division of the linear scale, which is set in the axis-specific MD **31010 ENC_GRID_POINT_DIST** is zero.

Reaction
NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Please inform the authorized personnel/customer service.
Enter a scale division of the linear scale according to the specifications of the machine manufacturer (or the measuring equipment manufacturer) in MD 31010 **ENC_GRID_POINT_DIST**.

Program continuation by
Power ON

26005 Axis %1 Parameterization error:Output evaluation

Explanation
%1 = axis name, spindle number

The output evaluation of the analog speed setpoint, which is set in MD 32250 **RATED_OUTVAL** or MD 32260 **RATED_VELO** is zero.

Reaction
NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.

Alarm display.
Interface signals are set.

Remedy: Enter the motor nominal speed in MD 32260 **RATED_VELO**.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

26006

Axis %1 encoder %2 Encoder type/output type %3 not possible

Explanation:

%1 = axis name, spindle number
%2 = encoder number
%3 = encoder type/output type

Not every encoder type or output type is possible in the current software version.

MD 30240 **ENC_TYPE**	= 0	Simulation
	= 1	Raw-signal encoder
	= 2	Square-wave encoder
	= 4	EnDat absolute encoder
MD 30130 **CTRLOUT_TYPE**	= 0	Simulation
	= 1	Default

Reaction:

NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy: Check MD 32240 **ENC_TYPE** and/or MD 30130 **CTRL_OUT_TYPE** and correct if necessary.

Program continuation by: Power ON

26014

Axis %1 MD %2 Illegal value

Explanation:

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

MD contains an invalid value.

Reaction:

NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy: Repeat your entry with the correct value and Power On.

Program continuation by: Power ON

26015

Axis %1 Machine data %2 [%3] Illegal value

Explanation:

%1 = axis name, spindle number
%2 = string: MD identifier
%3 = index: MD array index

MD contains an invalid value.

Reaction:

NC not ready.
NC Start inhibited.
NC Stop at alarm.

	The NC switches to the follow-up mode. Alarm display. Interface signals are set.
Remedy	Repeat your entry with the correct value and Power On.
Program continuation by	Power ON

26016 Axis %1 machine data %2 [%2] Illegal value

Explanation	%1 = axis name, spindle number %2 = string: MD identifier Machine data contains an invalid value.
Reaction	NC not ready. NC Start inhibited. NC Stop at alarm. The NC switches to the follow-up mode. Alarm display. Interface signals are set.
Remedy	Repeat your entry with the correct value and push RESET.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

26017 Axis %1 Machine data %2 [%3] Illegal value

Explanation	%1 = axis name, spindle number %2 = string: MD identifier %3 = index: MD array index Machine data contains an invalid value.
Reaction	Channel not ready. NC Start inhibited. NC Stop at alarm. The NC switches to the follow-up mode. Alarm display. Interface signals are set.
Remedy	Repeat your entry with the correct value and push RESET.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

26018 Axis %1 Setpoint output of drive %2 used repeatedly

Explanation	%1 = axis name, spindle number %2 = drive number The same setpoint assignment has been used several times. The machine data 30110 **CTRLOUT_MODULE_NR** contains the same value for different axes.
Reaction	NC not ready. NC Start inhibited. NC Stop at alarm. The NC switches to the follow-up mode. Alarm display. Interface signals are set.
Remedy	Please inform the authorized personnel/customer service. Avoid double assignment of the setpoint assignment by correcting the MD 30110 **CTRLOUT_MODULE_NR.**

Program continuation by: Power ON

26020

Axis %1 encoder %2 Hardware error %3 reinitializing encoder

Explanation

%1 = axis name, spindle number
%2 = encoder number
%3 = error fine coding

Error when initializing or accessing the encoder (cf. additional information for the absolute encoder interface in the error fine identification).

Reaction

NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference point).

Remedy

Please inform the authorized personnel/customer service.
Eliminate the hardware error; if necessary change the encoder.

Bit No.	**Meaning**	**Note**
Bit 0	Failure of illumination	
Bit 1	Signal amplitude too low	
Bit 2	False position value	
Bit 3	Overvoltage	
Bit 4	Undervoltage	
Bit 5	Overcurrent	
Bit 6	Battery change required	
Bit 7	Control check error	
Bit 8	EnDat encoder: incorrect overlap	
Bit 9	C/D track for ERN1387 encoder faulted, or EQN encoder connected or not correctly parameterized (not to EQN, MD 1011)	
Bit 10	Protocol cannot be interrupted or old HW	
Bit 11	SSI level identified on the data line or no encoder connected or incorrect encoder cable (ERN instead of EQN)	
Bit 12	TIMEOUT when reading measured values	
Bit 13	CRC error	
Bit 14	Incorrect IPU submodule for direct measuring signal	Only for 611D expansion
Bit 15	Encoder defective	

Program continuation by: Power ON

26022 Axis %1 encoder %2 Measuring with simulated encoder not possible

Explanation

%1 = NC axis number
%2 = encoder number

This alarm will occur at the control system when measuring without encoder hardware (simulated encoder).

Reaction

Alarm display
Interface signals are set.
NC Stop at alarm.
NC not ready.

Remedy

Please inform the authorized personnel/customer service.

- If possible modify the measuring movement such that the axis concerned need not traverse, and do not program this axis in the MEAS block. In this case, however, a measured value for this axis can also no longer be requested.
- Make sure that the measuring operation is not carried out using simulated encoders
  (MD 30240 **ENC_TYPE**).

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

26025 Axis %1 machine data %2 Value adapted

Explanation

%1 = axis name, spindle number
%2 = string: MD identifier
%3 = index: MD array index

The machine data contained an invalid value. It was therefore changed by the software internally to a valid value.

Reaction

Alarm display.

Remedy

Check the MD.

Program continuation by

Use the RESET key to cancel the alarm. Restart the part program.

26030 Axis %1 encoder %2 Absolute position lost

Explanation

%1 = axis name, spindle number %2 = encoder number

The absolute position of the absolute encoder became invalid, since a modified gear stage transmission ratio was detected between encoder and machining.

Reaction

NC not ready.
Under certain circumstances, it can be switched across all channels via MD.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference point).

Remedy

Please inform the authorized personnel/customer service.
Rereference/resynchronize the absolute encoder; mount the absolute encoder on the load side; configure it correctly (e.g. MD 31040 **ENC_IS_DIRECT**).

Program continuation by

Use the RESET key to cancel the alarm.

26050 Axis %1 Parameter set change from %2 to %3 not possible

Explanation
%1 = axis name, spindle number
%2 = index: current parameter set
%3 = index: New parameter record

The parameter set change cannot be carried out without steps. The cause is to be found in the contents of the parameter record to be enabled, e.g. different load gear factors.

Reaction
NC Start inhibited.
NC Stop at alarm.
Alarm display.
The NC switches to the follow-up mode.
Interface signals are set.
Local reaction

Remedy
-

Program continuation by
Use the RESET key to cancel the alarm. Restart the part program.

Use the "Cancel" key to cancel the alarm. No further operation required.

26052 Channel %1 in block %2: Path velocity for auxiliary function output too high

Explanation
%1 = channel number
%2 = block number, label

The alarm usually occurs in a block with auxiliary function output during the movement. In this case, the control system had to wait for the acknowledgment of the auxiliary function longer than planned. This alarm will also occur if unexpected control-internal inconsistencies block the continuous-path control mode (G64, G641, ...).
The path interpolation will suddenly stop at the signaled block end (generatory stop). With the next block change, traversing will go on unless the sudden stop has not caused an error on the position controller (e.g. by a very sensitively set MD 36400 **CONTOUR_TOL**).

Reaction
Alarm display.

Remedy
Program G09 in the signaled block to make sure that the path interpolation stops at the block end as scheduled.

Program continuation by
Use the "Cancel" key to cancel the alarm. No further operation required.

26100 Axis %1, drive %2 Sign of life failure

Explanation
%1 = axis name, spindle number
%2 = drive number

The drive control increments a sign-of-life cell in each control cycle, which is checked for a change by the servo in the interpolation cycle. If the cell remains unchanged, the alarm will be set.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Together with this alarm, further error messages (system errors, e.g. stack overflow) are displayed whose evaluation provides a conclusion to the error cause.

If this alarm occurs repeatedly, all the following alarm displays should be laid down in the form of a machine and program analysis, which should be as exact as possible, and be reported to Siemens AG, System Administration for A&DMC Products, hotline (tel. see page 9).

Program continuation by
Power ON

26101 Axis %1, drive %2 does not communicate

Explanation
%1 = axis name, spindle number
%2 = drive number

The drive does not communicate.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference points).

Remedy
Check the bus configuration.
Check connection (plug/connector disconnected, option module inactive etc.).

Program continuation by
Use the RESET key to cancel the alarm.

26102 Axis %1, drive %2 Sign of life failure

Explanation
%1 = axis name, spindle number
%2 = drive number

The sign-of-life cell is no longer updated by the drive.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.
The axes are no longer synchronized with the machine actual value (reference points).

Remedy
Check the clock settings; if necessary extend the cycle time.
Restart drive, check drive software.

Program continuation by
Use the RESET key to cancel the alarm.

26105 Drive for axis %1 not found

Explanation
%1 = axis name, spindle number

The drive parameterized for the specified axis could not be found. For example, a Profibus slave was parameterized in the NC which is not contained in the SDB1000.

Reaction
NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.

Alarm display.
Interface signals are set.

Remedy

Possible causes are:

1. By mistake, MD 30130 **CTRLOUT_TYPE** is unequal to 0; originally, it was intended to simulate the drive (=0).
2. MD 30110 **CTRLOUT_MODULE_NR** has been entered incorrectly, i.e. the logic drive numbers have been changed, or a drive number has been entered which does not exist at the bus (it is recommended to check, e.g. the number of slaves)
3. You are using a wrong SDB1000 or you have not selected identical addresses of input and output slots of the drives.

Program continuation by

Power ON

26106

Encoder 2 for axis %1 not found

Explanation

%1 = axis name, spindle number
%2 = encoder number

The drive parameterized for the specified axis could not be found. For example, a Profibus slave was parameterized in the NC which is not contained in the SDB1000.

Reaction

NC not ready.
NC Start inhibited.
NC Stop at alarm.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy

Possible causes are:

1. By mistake, MD 30240 **ENC_TYPE** is unequal to 0; originally, it was intended to simulate the encoder (=0).
2. MD 30220 **ENC_MODULE_NR** has been entered incorrectly, i.e. the logic drive numbers have been changed, or a drive number has been entered which does not exist at the bus (it is recommended to check, e.g. the number of slaves)
3. You are using a wrong SDB1000 or you have not selected identical addresses of input and output slots of the drives.

Program continuation by

Power ON

300402

System error in drive coupling.Error codes %1, %2

Explanation

%1 = error code 1
%2 = error code 2

An internal software error or a fatal error status has occurred which can possibly be corrected by a hardware reset. The cause can generally only be found by the System Administration for A&DMC Products of Siemens AG, hotline (tel.: see page 11).

In case of the error combination (1077,X), the assignment of computational time for the drive communication subtask should be increased via MD10140 **TIME_LIMIT_NETTO_DRIVE_TASK** (possible up to 500ms).

If the above mentioned limit is exhausted and the alarm persists, MD10150 MN**PREP_DRIVE_TASK_CYCLE_RATIO=1** can be additionally set. When doing so, make sure that a reduction of MD10150 does not result in a reduction of the time portion in the preparation in the non-cyclic time level, what, in turn, may result in larger block cycle times.

Reaction
NC not ready.
NC Stop at alarm.
NC Start inhibited.
The NC switches to the follow-up mode.
Alarm display.
Interface signals are set.

Remedy
Report your error text to Siemens AG A&DMC, hotline
(tel.: see page 11).

Program continuation by
Power ON

300410

Axis %1, drive %2 Error saving file (%3, %4)

Explanation
%1 = NC axis number
%2 = drive number
%3 = error code 1
%4 = error code 2

A data block, e.g. the result of a measuring function, could not be saved in the file system.

Reaction
Alarm display.
Interface signals are set.

Remedy
Provide for more space in the file system. In most cases, it is sufficient to delete 2 NC programs or to provide for 4 - 8 KB more space.

Program continuation by
Use the RESET key to cancel the alarm.

300411

Axis %1, drive %2 Error reading file (%3, %4)

Explanation
%1 = NC axis number
%2 = drive number
%3 = error code 1
%4 = error code 2

A data block, e.g. a drive boot file, could not be read from the file system. The data record or the file system is damaged.

Reaction
Alarm display.
Interface signals are set.

Remedy
If the error occurred on booting, i.e. pertains to a drive boot file, delete the boot files and reload them from a data backup into the control system.

Program continuation by
Use the RESET key to cancel the alarm.

300412

Error saving file (%1, %2)

Explanation
%1 = error code 1
%2 = error code 2

A data block, e.g. the result of a measuring function, could not be saved in the file system.

Reaction
Alarm display.
Interface signals are set.

Remedy	Please inform the authorized personnel/customer service. Provide for more space in the file system. In most cases, it is sufficient to delete 2 NC programs or to provide for 4 - 8 KB more space.
Program continuation by	Use the RESET key to cancel the alarm.

300413

Error reading file (%1, %2)

Explanation	%1 = error code 1 %2 = error code 2 A data block, e.g. a drive boot file, could not be read from the file system. The data record or the file system is damaged.
Reaction	Alarm display. Interface signals are set.
Remedy	If the error occurred on booting, i.e. pertains to a drive boot file, delete the boot files and reload them from a data backup into the control system.
Program continuation by	Use the RESET key to cancel the alarm.

300423

Measurement result could not be read (%1)

Explanation	%1 = error code The result of a measuring process could not be read: Error code = 4:Not enough space for the measurement result Error code = 16: Measurement not yet completed
Reaction	Alarm display. Interface signals are set.
Remedy	Repeat the measurement. If necessary change the measuring time.
Program continuation by	Use the RESET key to cancel the alarm.

300500

Axis %1, drive %2 System error drive, error codes %3, %4

Explanation	%1 = NC axis number %2 = drive number %3 = error code 1 %4 = error code 2 The drive is reporting a system error.
Reaction	NC not ready. Under certain circumstances, it can be switched across all channels via MD. Channel not ready. NC Stop at alarm. NC Start inhibited. The NC switches to the follow-up mode. Alarm display. Interface signals are set.
Remedy	For notes with regard to the error codes, please refer to Section 1.7 "Error codes of alarm 300500" at the end of this publication. NC RESET (POWER ON) A more detailed search for the error cause can only be done by the developer team. To this aim, it is imperative to specify the error codes displayed. Reinitialize the drive. Please inform the authorized personnel/customer service. Siemens AG, System Administration for A&D MC Products, hotline (tel.: see Section 1.1).

Program continuation by Power ON

Reaction With feed drives: Generatory stop (corresponds to STOP B)
With main drives: Pulse inhibit and servo disable (corresponds to STOP A).

Explanation This error occurs if the computational time of the drive processor is no longer sufficient for the clock cycle specified in the additional information.

Error No.	Additional Information	Explanation
...		
With 840D: 03 With 840C: F003	40	Monitoring cycle for SINUMERIK Safety Integrated too small
...		

Remedy Increase either the appropriate clock or the underlying clock (e.g. current, speed, position control clock) or deselect functions not required.

1.2 Profibus Alarms

380001

Profibus DP:Boot error, cause %1 parameters %2 %3 %4

Explanation

%1 = error cause
%2 = parameter 1
%3 = parameter 2
%4 = parameter 3

Error on booting Profibus-DP Master.

Error cause:	Par 1:	Par 2:	Par 3:
01=DPM version	DPM version	DPA version	
02=DPM boot time-out	DPM actual status	DPM set status	
03=DPM boot status	DPM actual status	DPM set status	DPM error code
04=DPM boot error	DPM actual status	DPM set status	DPM error code
05=DPM-PLL sync error			
07=alarm source too long	Actual number	Setpoint number	
08=unknown client	Client ID		
09=client version	Client ID	Client version	DPA version
10=too many clients	Client number	Max. number of clients	

Clients are the following components of the control system, which use Profibus DP:
Client ID = 1: PLC
Client ID = 2: NCK

Possible causes are:
- SDB1000 contains faulty data
- Hardware of NC component defective

Reaction

Alarm display
Interface signals are set.
NC not ready.
NC start inhibited.

Remedy

Please carry out the following procedure:

1. Check the control project (in particular, SDB1000), check MD 11240; if you use a user-specific SDB1000, reload it.
2. If the error persists, save the file and restart the control system with the factory-set default values.
3. If the control system boots without errors, the user data should be re-loaded step by step.
4. If the error also persists when rebooting the control system with the default values, either reboot the control system from the PC Card or carry out a software update.

5. If the error nevertheless persists, change the hardware. If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.

Program continuation by: Power ON

380003

Profibus DP:Fault, cause %1, R parameters %2 %3 %4

Explanation

%1 = error cause
%2 = parameter 1
%3 = parameter 2
%4 = parameter 3

A fault has occurred at Profibus DP during cyclic operation.

Error cause:	Par 1:	Par 2:	Par 3:
01=unknown alarm	Alarm class	Logical address	
02=DPM cycle time-out	DPM actual status	DPM set status	
03=DPM cycle status	DPM actual status	DPM set status	DPM error code
04=DPM cycle error	DPM actual status	DPM set status	DPM error code
05=not registered Client	Client number	Max. number of clients	

Alarm class: (compare with alarm 380 060)

Predominantly, in case of error cause 01, possible causes can be:

- Data transfer on Profibus DP disturbed

In case of error causes 02, 03, 04:

- SDB1000 contains false data

In case of error causes 02, 03, 04, 05:

- Parts of the system program have been damaged

Reaction

Alarm display
Interface signals are set.
NC not ready.
NC start inhibited.

Remedy

In case of error cause 01:

- Check whether the electrical standards and instructions regarding faults with regard to Profibus DP are observed; check the cable routing.
- Check the terminators of the Profibus connectors (on the line ends - position CN; otherwise, position OFF prescribed)
- Check the slave in case of the error causes 02, 03, 04:
- Check SDB1000 in case of the error causes 02,03, 04, 05:
- To carry out the troubleshooting, proceed as with alarm 380001.

If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.

Program continuation by: Use the RESET key to cancel the alarm.

380020

Profibus DP: SDB1000 error %1 for SDB source %2

Explanation

%1 = error cause
%2 = SDB1000 source

SDB1000 for configuring Profibus DP faulty.

Error cause:

01=SDB1000 in SDB1000 source does not exist
02=SDB1000 in SDB1000 source too large.
03=SDB1000 in SDB1000 source cannot be activated.

SDB1000 source:

00=Default SDB1 (selection by MD 11240 = 0, if no user SDB1000 loaded in the control system)

01=default SDB1 (selection by MD 11240 = 1)
02=default SDB2 (selection by MD 11240 = 2)

...

100 = SDB1000 contained in battery-backed memories (SRAM)
101 = user SDB1000 contained in the file system
102 = SDB1000 loaded in the SRAM on booting

Reaction

Profibus is inactive or operates as per default SDB1000
Alarm display,
interface signals are set,
NC not ready,
NC Start inhibited

Remedy

- Check MD 11240.
- In the case SDB1000 source = 100: Reload the user SDB1000 into the passive file system /_N_IBN_DIR/_N_SDB1000_BIN.
- In case of SDB1000 source = 101: Check the backup batteries.
- In the case SDB1000 source = 102: To carry out the troubleshooting, proceed as with alarm 380 001.
- If alarm 380 021 is additionally signaled, proceed according to the provided notes.

If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.

Program continuation by

Power ON

380021

Profibus DP:Default SDB1000 has been loaded.

Explanation

No user-specific SDB1000 exists.
Default SDB1000 was loaded on booting.

The NC can be started up without process I/O.

This alarm occurs when the NC is turned on for the first time or once when the SDB1000 stored in the battery-backed RAM is lost.

Reaction

Alarm display.

Remedy

Create the application-specific SDB1000 and load it into the control system or select and activate the default SDB1000 via MD 11240 **PROFIBUS_SDB_NUMBER**.

Restart the NC.

If the error occurs again when the NC is turned on the next time, the loaded SDB1000 is faulty and a new SDB1000 must be created.

Program continuation by: Use the "Cancel" key to cancel the alarm.

380040

Profibus DP:Configuration error %1, parameter %2

Explanation: %1 = error cause
%2 = parameter

Profibus DP in SDB1000 was not created according to the Planning Guide of the NC used.

Error cause:	Par 1:
01 = SDB1000 contains slave or diagnostics slot	Slave address
02 = SDB 1000 contains too many slot entries	Identifier

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

Remedy: Check whether SDB1000

- contains a diagnostic slot for each slave and
- contains only application-relevant slave entries.

In principle, it is possible to include a superset of slaves in SDB1000, which are partially relevant for different end variants of the product. This, however, results in overloading the NC with reference to memory and runtime requirements and should therefore generally be avoided. If this alarm occurs, SDB1000 should be reduced.

If the alarm persist, please consult the control system manufacturer, sending him the error text.

Program continuation by: Power ON

380050

Profibus DP:Multiple assignment of inputs to address %1

Explanation: %1 = logical address

The assignment of the input data in the logical address range is assigned repeatedly.
Logical address: Base address of the address range defined several times

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

Remedy: Check the address space allocation as follows:

Check the following machine data for multiple assignment:

MD 13050[1] - MD 13050[n]	n = largest axis index of control system
MD 12970, 12971	PLC address range of digital inputs
MD 12978, 12979	PLC address range of analog outputs

If this parameterization does not contain any inconsistencies, compare these MD against the configuration in SDB1000. When doing so, make sure, in particular, that the individual ranges do not overlap, resulting from the configured slot lengths. After locating the error cause, modify the MD and/or SDB1000.

Program continuation by: Power ON

380051

Profibus DP:Multiple assignment of outputs to address %1

Explanation

%1 = logical address

The assignment of the input data in the logical address range is assigned repeatedly.
Logical address: Base address of the address range defined several times

Reaction

Alarm display
Interface signals are set.
NC not ready.
NC start inhibited.

Remedy

Check the address space allocation as follows:

Check the following machine data for multiple assignment:

MD 13050[1] - MD 13050[n]	n = largest axis index of control system
MD 12974, 12975	PLC address range of digital inputs
MD 12978, 12979	PLC address range of analog outputs

If this parameterization does not contain any inconsistencies, compare these MD against the configuration in SDB1000. When doing so, make sure, in particular, that the individual ranges do not overlap, resulting from the configured slot lengths. After locating the error cause, modify the MD and/or SDB1000.

Program continuation by

Power ON

380060

Profibus DP:Alarm %1 on logical address %2 from station not assigned

Explanation

%1 = alarm class
%2 = logical address

SDB1000 contains a slave that is not assigned in the NC by MD parameterization (see Remedy for alarm 380 050/51). The slave is also connected to Profibus DB. Such a slave caused an alarm.

Alarm class:

01 = station recurrence (or coming)
02 = station failure

Working with the NC is possible.

Reaction

Alarm display.

Remedy

- Add the required MDs or
- modify SDB1000 or
- disconnect slave from Profibus DP or
- acknowledge the alarm.

Program continuation by

Use the "Cancel" key to cancel the alarm.

380070

Profibus DP:No input slot exists for the base address %1 (length %2)

Explanation

%1 = logic base address of the requested range
%2 = size of range in bytes

An invalid base address has been defined for a digital input.

Either no configured slot at all exists for this base address, or the requested range protrudes the end of the slot.

Reaction

Alarm display
Interface signals are set.
NC not ready
NC Start inhibited

Remedy	Check your hardware configuration for modules not plugged or defective and replace them. If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.
Program continuation by	Power ON

380071

Profibus DP:No output slot exists for the base address %1 (length %2)

Explanation	%1 = logic base address of the requested range %2 = size of range in bytes An invalid logical base address has been defined for a digital or analog input. Either no configured slot at all exists for this base address, or the requested range protrudes the end of the slot.
Reaction	Alarm display Interface signals are set. NC not ready NC Start inhibited
Remedy	Check your hardware configuration for modules not plugged or defective and replace them. If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.
Program continuation by	Power ON

380072

Profibus DP:Illegal output slot base address %1 (size %2)

Explanation	%1 = logic base address of the requested range %2 = size of range in bytes An invalid logical base address has been defined for a digital or analog output; the range is in the access area of the PLC (process output map, base addresses < 128).
Reaction	Alarm display Interface signals are set. NC not ready NC Start inhibited
Remedy	Check your hardware configuration for modules not plugged or defective and replace them. If the error cannot be eliminated using procedure described above, contact the control system manufacturer, sending him the error text.
Program continuation by	Power ON

380075

Profibus DP:Failure DP I/Os slave %1

Explanation	%1 = slave address Failure of a Profibus slot used by the NCK for digital or analog I/Os
Reaction	Alarm display.
Remedy	Check whether the Profibus slave functions correctly (make sure that all slaves are included into the bus - green LED).
Program continuation by	The alarm display will disappear with the alarm cause. No further operation required.

380500

Profibus DP:Fault drive %1, code %2, value %3, time %4

Explanation	%1 = axis %2 = fault code of the drive (P824) %3 = fault value of the drive (P826) %4 = fault time of the drive (P825) Contents of the fault memory of the assigned drive
Reaction	Alarm display.
Remedy	For the fault codes/fault values, see drive documentation.
Program continuation by	The alarm display will disappear with the alarm cause. No further operation required.

1.3 Cycle Alarms

60000 Channel %1 block %2

Explanation	%1 = channel number %2 = block number, label
Reaction	Alarm display. Interface signals are set. NC Start inhibited.
Remedy	-
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61000 No tool offset active

Source (cycle)	SLOT1, SLOT2 POCKET3, POCKET4 CYCLE71 CYCLE72 CYCLE93 to CYCLE95
Reaction	Block preparation in NC is canceled
Remedy	Program a tool with tool offset in the called program.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61001 Incorrect pitch

Source (cycle)	CYCLE84 CYCLE840 CYCLE97 CYCLE376T
Reaction	Block preparation in NC is canceled
Remedy	Check the parameter for the thread size or check lead specification (are not consistent)
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61002 Type of machining incorrectly defined

Source (cycle)	SLOT1, SLOT2 POCKET3, POCKET4 CYCLE71 CYCLE72 CYCLE93 CYCLE95 CYCLE97
Reaction	Block preparation in NC is canceled
Remedy	The value of parameter VARI for the machining type is incorrectly set and must be altered.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61003 No feed programmed in the cycle

Source (cycle)	CYCLE71 CYCLE72 CYCLE371T to CYCLE374T CYCLE383T to CYCLE385T CYCLE381M, CYCLE383M, CYCLE384M, CYCLE387M
Reaction	Block preparation in NC is canceled
Remedy	The parameter for the feed is incorrectly specified and must be modified.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61009 Active tool number = 0

Source (cycle)	CYCLE71 CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	No tool (T) is programmed prior to the cycle call.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61010 Finishing allowance too large.

Source (cycle)	CYCLE71 CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	A scaling factor is active, what is not permissible for the cycle.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61011 Illegal scaling

Source (cycle)	CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	The finishing allowance at the root is larger than the overall depth; it must be reduced.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61101 Reference plane incorrectly defined

Source (cycle)	CYCLE71 CYCLE22 CYCLE81 to CYCLE88 CYCLE840 SLOT1, SLOT2 POCKET3, POCKET4
Reaction	Block preparation in NC is canceled
Remedy	Either choose different values for reference and retraction planes when specifying the depth directly, or specify an absolute value for the depth.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61102 No spindle direction programmed

Source (cycle)	CYCLE86 CYCLE88 CYCLE840 CYCLE370T to CYCLE374T, CYCLE376T CYCLE383T to CYCLE385T CYCLE381M, CYCLE383M, CYCLE384M, CYCLE387M POCKET3, POCKET4
Reaction	Block preparation in NC is canceled
Remedy	Parameter SDIR (or SDR in CYCLE840) must be programmed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61103 Number of holes is zero

Source (cycle)	HOLES1 HOLES2
Reaction	Block preparation in NC is canceled
Remedy	No value for the number of drill holes programmed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61104 Contour violation of grooves/slots

Source (cycle)	SLOT1 SLOT2
Reaction	Block preparation in NC is canceled
Remedy	Faulty parameterization of the milling pattern in the parameters that define the position of the grooves/slots on the circle and their form
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61105 Milling cutter radius too large

Source (cycle)	SLOT1, SLOT2 POCKET3, POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The milling cutter radius in the tool offset memory is greater than the pocket or groove width. Use smaller cutter or modify pocket width.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61106 Number ordistance of circle elements too large

Source (cycle)	HOLES2 SLOT1, SLOT2
Reaction	Block preparation in NC is canceled
Remedy	Faulty parameterization of NUM or INDA The arrangement of the circle elements on a full circle is not possible.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61107 First drilling depth incorrectly defined

Source (cycle)	CYCLE83
Reaction	Block preparation in NC is canceled
Remedy	Modify the value for the first drilling depth (first drilling depth is opposite to the total drilling depth).
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61108 No permissible values for parameters _RAD1 and _DP1

Source (cycle)	POCKET3 POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The parameters _RAD1 and _DP for defining the path for depth infeed were incorrectly specified.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61109 Parameter _CDIR incorrectly defined

Source (cycle)	POCKET3 POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The value of the parameter defining the milling direction _CDIR was incorrectly specified and must be modified.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61110 Finishing allowance on the root > depth infeed

Source (cycle)	POCKET3 POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The finishing allowance on the root has been specified greater than the maximum depth infeed; either reduce the finishing allowance or increase the depth infeed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61111 Infeed width > tool diameter

Source (cycle)	CYCLE71 POCKET3 POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The programmed infeed width is greater than the diameter of the active tool; it must be reduced.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61112 Tool radius negative

Source (cycle)	CYCLE72
Reaction	Block preparation in NC is canceled

Remedy	The radius of the active tool is negative; this is not permissible.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61113 Parameter _CRAD for corner radius too large

Source (cycle)	POCKET3
Reaction	Block preparation in NC is canceled
Remedy	The parameter for the corner radius _CRAD was specified too large; it must be reduced.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61114 Direction of machining G41/G42 incorrectly defined

Source (cycle)	CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	The direction of machining of the cutter radius compensation G41/G42 was incorrectly programmed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61115 Approach and retraction mode (straight line/circle/plane/space) incorrectly defined.

Source (cycle)	CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	The mode of approaching to or retracting from the contour has been incorrectly defined; check parameter _AS1 or _AS2.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61116 Approach or retraction travel=0

Source (cycle)	CYCLE72
Reaction	Block preparation in NC is canceled
Remedy	The approach or retraction travel is specified with zero; it must be increased; check parameter _LP1 or _LP2.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61117 Active tool radius <=0

Source (cycle)	CYCLE71 POCKET3 POCKET4
Reaction	Block preparation in NC is canceled
Remedy	The radius of the active tool is negative or zero; this is not permissible.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61118 Length or width =0

Source (cycle)	CYCLE71

Reaction	Block preparation in NC is canceled
Remedy	The length or width of the milling face is not permissible; check the parameters _LENG and _WID.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61124 Infeed width not programmed

Source (cycle)	CYCLE71
Reaction	Block preparation in NC is canceled
Remedy	With the simulation active without tool, a value for the infeed width _MIDA must always be programmed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61601 Finished part diameter too small

Source (cycle)	CYCLE94
Reaction	Block preparation in NC is canceled
Remedy	A finished-part diameter < 3mm has been programmed. Increase value.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61602 Tool width incorrectly defined

Source (cycle)	CYCLE93
Reaction	Block preparation in NC is canceled
Remedy	The tool width (recessing tool) is greater than the programmed the programmed groove width.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61603 Groove shape incorrectly defined

Source (cycle)	CYCLE93 CYCLE374T
Reaction	Block preparation in NC is canceled
Remedy	⇒ Radii/chamfers on recess base do not match with groove width ⇒ Face groove not possible on a contour element running parallel to the longitudinal axis.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61604 Active tool violates the programmed contour

Source (cycle)	CYCLE95
Reaction	Block preparation in NC is canceled
Remedy	Contour violation in relief cut elements due to clear cutting angle of the tool used, i.e. use a different tool or check the contour subroutine.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61605 Contour incorrectly programmed

Source (cycle)	CYCLE95

Reaction	Block preparation in NC is canceled
Remedy	Illegal relief cut element found
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61606 Error in contour preparation

Source (cycle)	CYCLE95
Reaction	Block preparation in NC is canceled
Remedy	Check contour subroutine This alarm always occurs in conjunction with NCK alarms 10930 ... 10934, 15800 or 15810.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61607 Starting point incorrectly defined

Source (cycle)	CYCLE95 CYCLE376T
Reaction	Block preparation in NC is canceled
Remedy	The start point reached prior to the cycle call is not outside the rectangle described by the contour subroutine.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61608 Wrong tool point direction programmed

Source (cycle)	CYCLE94
Reaction	Block preparation in NC is canceled
Remedy	An edge position 1 .. 4 matching with the recess shape must be programmed.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61609 Shape incorrectly defined

Source (cycle)	CYCLE94
Reaction	Block preparation in NC is canceled
Remedy	Check the parameters for the recess shape.
Program continuation by	Use the RESET key to cancel the alarm. Restart the part program.

61610 No infeed depth programmed

Source (cycle)	CYCLE374T
Remedy	Change the infeed depth.

61611 No intersection point found

Source (cycle)	CYCLE95
Reaction	Block preparation in NC is canceled
Remedy	No intersection point with the contour could be calculated. Check contour programming or change infeed depth.

Program continuation by: Use the RESET key to cancel the alarm. Restart the part program.

61800

Channel %1 block %2: Ext. CNC system missing

Source (cycle): CYCLE328,
CYCLE370T to CYCLE374T, CYCLE376T,
CYCLE383T to CYCLE385T,
CYCLE381M, CYCLE383M, CYCLE384M,
CYCLE387M

Remedy: Set the machine data for an external language MD 18800 **MM_LANGUAGE** or the option bit 19800 **ON_EXTERN_LANGUAGE.**

61801

Channel %1 block %2: Wrong G code selected

Source (cycle): CYCLE370T to CYCLE374T, CYCLE376T,
CYCLE383T to CYCLE385T

Explanation/remedy: An illegal numerical value was programmed in the program call CYCLE ... <value>, or an invalid value was specified in the cycle setting data for the G code system. Correct the values.

61802

Channel %1 block %2: Wrong axis type

Source (cycle): CYCLE328

Explanation/remedy: The programmed axis is assigned to a spindle.

61803

Channel %1 block %2: Programmed axis does not exist

Source (cycle): CYCLE328

Explanation/remedy: The programmed axis does not exist in the system. Check MD 20050 ... MD20080.

61804

Channel %1 block %2: Progr. position exceeds reference point

Source (cycle): CYCLE328

Explanation/remedy: The programmed intermediate position or current position is after the reference point.

61805

Channel %1 block %2: Value programmed absolutely or incrementally

Source (cycle): CYCLE328
CYCLE371T to CYCLE374T, CYCLE376T,
CYCLE383T, CYCLE384T

Explanation/remedy: The programmed intermediate position is programmed both absolutely and incrementally.

61806

Channel %1 block %2: Wrong axis assignment

Source (cycle): CYCLE328

Explanation/remedy: The order of sequence of the axis is wrong.

61807 **Channel %1 block %2: Wrong spindle direction programmed (active)**

Source (cycle) CYCLE384M

Explanation/remedy The programmed spindle direction is in conflict with the spindle direction intended for the cycle.

61808 **Channel %1 block %2: Final drilling depth or single drilling depth missing**

Source (cycle) CYCLE381M, CYCLE383M, CYCLE384M,
CYCLE387M,
CYCLE383T to CYCLE385T

Explanation/remedy Total depth "Z" or single drilling depth "Q" missing in the G8x block (initial call)

61809 **Channel %1 block %2: Illegal drilling position**

Source (cycle) ISO shell cycles

61810 **Channel %1 block %2: ISO G code not possible**

Source (cycle) ISO shell cycles

61811 **Channel %1 block %2: Illegal ISO axis name**

Source (cycle) CYCLE370T to CYCLE374T,
CYCLE376T,
CYCLE383T to CYCLE385T

Explanation/remedy An illegal numerical value was defined in the calling block.

61812 **Channel %1 block %2: Value(s) in external cycle call incorrectly defined**

Source (cycle) CYCLE370T to CYCLE374T,
CYCLE376T

Explanation/remedy An illegal ISO axis name was programmed in the calling block.

61813 **Channel %1 block %2: GUD value incorrectly defined**

Source (cycle) CYCLE376T

Explanation/remedy An illegal numerical value was entered in the cycle setting data.

61815 **Channel %1 block %2: G40 not active**

Source (cycle) CYCLE374T,
CYCLE376T

Explanation/remedy G40 was not active prior to the cycle call.

62000 **Channel %1 block %2**

Explanation %1 = channel number
%2 = block number, label

Reaction Alarm display.

Remedy -

Program continuation by	Use either the "Cancel" key or press NC START to cancel the alarm.

62100

No drilling cycle active

Source (cycle)	HOLES1 HOLES2
Reaction	Block preparation is aborted
Remedy	Before calling the drilling pattern cycle, no drilling cycle was called modally.
Program continuation by	Use the "Cancel" key to cancel the alarm. Press NC Start to continue the cycle.

63000

Channel %1 block %2

Explanation	%1 = channel number %2 = block number, label
Reaction	Alarm display.
Remedy	-
Program continuation by	Use either the "Cancel" key or press NC START to cancel the alarm.

1.4 ISO Alarms

10796

Illegal axis names used

Explanation

Illegal axis names were used in MD20060 **AXCONF_GEOAX_NAME_TAB[]** and MD2080 **AXCONF_CHANAX_NAME_TAB[]**.

Permissible axis names are:
Fanuc T: X, Y, Z, C
Fanuc M: X, X, Z, for the 4th axis A and C

Axis names consisting of several letters or of a letter and a number, as well as lowercase letters are not permitted. The axis names can be freely distributed over the axes; the 1st axis need not be assigned axis name X.

Reaction

Alarm display
Interface signals are set
NC Start inhibited

Remedy

Adapt the axis names in the MD 20060 **AXCONF_GEOAX_NAME_TAB[]** and MD2080 **AXCONF_CHANAX_NAME_TAB[]** accordingly.

Program continuation by

18200

Channel %1 block %2 Language switchover not possible%3

Explanation

%1 = channel number
%2 = block number, label
%3 = reason

It is currently not possible to switch to an external NC language due to reason (%3).

Reaction

Alarm display
Interface signals are set
NC Start inhibited
Interpreter stop

Remedy

In case of reason (%3) } 1 : Deselect the transformation and try to switch the language once more.

Program continuation by

Press NC START to cancel the alarm and continue the program.

18201

Channel %1 block %2 G10 parameter %3 not programmed

Explanation

%1 = channel number
%2 = block number, label
%3 = reason

The parameter P, R or L is not programmed for the G command.

Reaction

Alarm display
Interface signals are set
NC Start inhibited
Interpreter stop

Remedy

Add the missing parameter in the block.

Program continuation by

Press NC START to cancel the alarm and continue the program.

18202 Channel %1 block %2 Skip only possible at the beginning of the block

Explanation	%1 = channel number %2 = block number, label The skip character / is not written at the beginning of the block.
Reaction	Alarm display Interface signals are set NC Start inhibited Interpreter stop
Remedy	Remove the skip character or write it at the beginning of the block.
Program continuation by	Press NC START to cancel the alarm and continue the program.

1.5 PLC Alarms

400000

PLC Stop [type]

Explanation

PLC is not in cyclic operation. Traversing with machine is not possible.

[Type]: 1 Ready (user program has not been started)
2 Break (user program has been interrupted)
3 Error (further PLC alarm with PLC Stop exists)

Reaction

Alarm display.

Remedy

Correct the other PLC alarm;
Switch-on menu stands on PLC Stop,
or test user program.

Program continuation by

- POWER ON
- Select the Start-up menu to continue
- Start via Programming Tool PT 802

400002

System error [type]

Explanation

[Type] : Type number

This alarm refers you to internal error conditions, which in conjunction with the error number contained in the error message provide information on the error cause and the error location.

Reaction

PLC stop

Remedy

Please report this error with type number to Siemens.
For error cause refer to the specified software section in the displayed line number

Program continuation by

Power ON

400004

Code error:[string] network [no.]

Explanation

[String] : internal error code, block type
[No.] : Network number

The user program contains an operation which is not supported by the control system.

Reaction

PLC stop

Remedy

Modify the user program and reload it.

Program continuation by

Power ON

400005

Switch-On menu is on PLC Stop

Explanation

The user program is not executed.

Reaction

Alarm display.

Remedy

Power ON

Program continuation by

The alarm display will disappear with the alarm cause.

- Power ON
- Use the Start-up menu to continue.

400006

Retentive PLC data lost

Explanation

This can have the following causes:

- Operator action (e.g. PLC general reset, boot with default values)
- Operator action - booting with saved data, without saving the data first
- Back-up time exceeded

Reaction: Alarm display.

Remedy: Update the required data.

Program continuation by: Press CLEAR to clear the alarm.

400007

Operand error:[string] network [no.]

Explanation

[String] : block type
[No.] : network number

Reaction: PLC stop

Remedy: Check the displayed variable in the user program for address range violation, invalid data type and alignment errors.

Program continuation by: Power ON

400008

Programming tool version not compatible [version]

Explanation: This version is not compatible with the product stage of the control system.

Reaction: PLC stop

Remedy: Compile the user program with a compatible programming tool version and load it into the control system.

Program continuation by: Power ON

400009

Computation time overflow on PLC level:[string] network [no.]

Explanation

[String] : block type
[Nr.] : network number

Check the user program of the respectively displayed network.

Reaction: PLC stop

Remedy: Modify the user program.

Program continuation by: Power ON

400010

Arithmetic error in the user program:[type] [string] network [no.]

Explanation

Check the user program in the specified network.

Type 1: Division by zero with fixed-point arithmetics
Type 2: Floating point arithmetics

[String] Type number, block identifier
[No.] Network number

Reaction: PLC stop

Remedy: Modify the user program.

Program continuation by: Power ON

400011

Permitted number of subroutine levels exceeded:[string] network [no.]

Explanation

[String] : Block identifier
[No.] : Network number

Check the user program in the specified network.

Reaction: PLC stop

Remedy: Modify the user program.

Program continuation by: Power ON

400013

PLC user program faulty

Explanation: The PLC user program in the control system is defective or missing.

Reaction: PLC stop

Remedy: Reload the PLC user program.

Program continuation by: Power ON

400014

Profibus DP boot sequence disturbed, types 1 - 4

Explanation

Type 1: Profibus DP not booted
Type 2: Software version NC does not match with PLC software version
Type 3: Number of slots per function exceeded
Type 4: Profibus DP server not ready

Reaction: PLC stop

Remedy

Types1 to 3: Report the error to SIEMENS.

Type 4: Check the 802D PCU hardware or change it or check MD 11240.

Program continuation by: Power ON

400015

Profibus DP I/O defective:Log. addr.[x] bus addr./slot: [y/z]

Explanation

The PLC user program uses I/O addresses that do not exist.

[x] Logical I/O address
[y] Slave number
[z] Slot number

Error causes:

- No voltage provided for Profibus I/O modules
- Slave bus address incorrectly set
- Profibus link defective
- Active MD 11240 (SDB Profibus configuration) is incorrectly set

Reaction: PLC stop

Remedy: Eliminate the error with reference to the particular error cause.

Program continuation by: Power ON

1.6 Action List

No./Name	Explanation	Not allowed if	Remedy
1. INIT	Carry out init phase (the tasks are initialized after Power On)		
2. RESET	Carry out RESET (VDI signal: after Reset, mode group reset ('BAG RESET') or Power On)		
3. RESET_INITBLOCK	Activate Reset Init blocks (VDI signal: after Reset)		
4. PROG_END	Carry out RESET, end of program has been detected (NC block with M30)		
5. MODESWITCHTOA-PROGMODE	Mode change to program mode MDA or Automatic (VDI signal: BAG signal)	1. the channel is active (program running, block search, loading of machine data) 2. the program has already been started in another program mode. 3. a channel has left the mode group because of an interrupt. 4. Overstore or Digitize has been selected.	⇒ Press RESET to cancel the program or stop program (not on block search or machine data loading) ⇒ Use the RESET key to cancel the program ⇒ Press RESET to cancel the program or wait until interrupt is completed. ⇒ Deselect Overstore, Digitize
6. MODESWITCHTOSAVE-MODE	Automatic switching from an internal mode to the mode externally set (With TEACH_IN, after each Stop, it is tried to switch from the internal mode "AUTOMATIC, MDA" to TEACH_IN)		
7. MODESWITCHTOHAND-MODE	Mode change to a manual mode (VDI signal: JOG, TEACH_IN, REF)	1. nesting depth too high: Various events (e.g. interrupt) may interrupt execution of the current program. Depending on the particular event, ASUP programs are activated. These ASUP programs can be aborted in the same manner as the user program. For memory reasons, any nesting depth of the ASUP programs is not possible. 2. the channel is active (program running, block search, loading of machine data) 3. a channel has left the mode group because of an interrupt. 4. Overstore or Digitize has been selected.	⇒ Press RESET to cancel the program ⇒ Press RESET to cancel the program or stop program (not on block search or machine data loading) ⇒ Press RESET to cancel the program or wait until the interrupt is completed. ⇒ Deselect Overstore, Digitize
8. OVERSTOREON	Selection of Overstore mode (PI command).		
9. OVERSTOREOFF	Deselection of Overstore mode (PI command).		

No./Name	Explanation	Not allowed if	Remedy
10. INTERRUPT	Carry out a user interrupt of the type "ASUP" (VDI signal, digital-analog interface, ASUP interface)	1. the channel is active because of block search or machine data loading 2. the channel is stopped and the ASUP "ASUP_START_MASK" must be started and the current block cannot be reorganized. 3. Digitalize is selected 4. reference-point approach has not yet been carried out. 5. in case of a brake Reorg error	⇒ Wait until block search or loading of machine data is completed, or press RESET to abort the program ⇒ Activate block change until the NC block can be reorganized ⇒ Deselect Digitalize ⇒ Carry out reference-point approach or ignore this state using the machine data "ASUP_START_MASK. ⇒ Cancel the program.
11. INTERRUPTFASTLIFT-OFF	Carry out an ASUP user interrupt with rapid lift (VDI signal, ASUP interface, digital-analog interface).	See 10	
12. INTERRUPTBLSYNC	Carry out an ASUP user interrupt at block end (VDI signal, ASUP interface, digital-analog interface)	See 10	
13. FASTLIFTOFF	Carry out a fast retraction		
14. TM_MOVETOOL	Move tool (with tool management only) (PI command)		
15. DELDISTOGO_SYNC	Carry out "Delete distance to go" or axis synchronization (VDI signal: Delete distance to go or follow-up mode) e.g., when starting axis control	1. nesting depth too high: 2. brake reorg error.	⇒ Cancel the program. ⇒ Cancel the program.
16. PROGRESETREPEAT	Cancel the subroutine repetition (VDI signal: Delete number of subroutine passes)	1. nesting depth too high: 2. brake reorg error.	⇒ Cancel the program. ⇒ Cancel the program.
17. PROGCANCELSUB	Cancel the subroutine execution. (VDI signal: program level abortion)	1. nesting depth too high: 2. brake reorg error.	⇒ Cancel the program. ⇒ Cancel the program.
18. SINGLEBLOCKSTOP	Activate single block (VDI signal: Activate single block)		
19. SINGLEBLOCKOFF	Switch off single block. (VDI signal: Activate single block)		
20. SINGLEBLOCK_IPO	Activate main run single block. Activate OPI variable and VDI signal: Activate single block)		
21. SINGLEBLOCK_ DECODIER	Activate decoding single block. Activate OPI variable and VDI signal: Activate single block)	1. nesting depth too high: 2. brake reorg error.	⇒ Wait until the preceding Asup is completed, or abort program ⇒ Cancel the program.
22. SINGLEBLOCK_ MAINBLOCK	Activate main program single block. Activate OPI variable and VDI signal: Activate single block)		
23. SINGLEBLOCK_PATH	Activate "traversing single block". Activate OPI variable and VDI signal: Activate single block)		

No./Name	Explanation	Not allowed if	Remedy
24. STARTPROG	Start program execution (VDI signal: NC Start)	1. program state active, 2. an alarm response is provided which prevents start or forces deceleration. 3. reference-point approach not yet carried out	⇒ - ⇒ Carry out alarm clear condition ⇒ Approach reference point
25. CHANNELSTARTPROG	Start program execution (channel communication, NC block: Start)	1. program state active, 2. an alarm response is provided which prevents start or forces deceleration. 3. reference-point approach not yet carried out 4. inappropriate operating mode selected (only Automatic mode)	⇒ Use WAIT to secure Start ⇒ Carry out alarm clear condition ⇒ Approach reference point ⇒ Select program mode
26. RESUMEPROG	Start continuation of program execution (VDI signal: NC Start)	1. program state active, 2. an alarm response is provided which prevents start or forces deceleration. 3. reference-point approach not yet carried out.	⇒ - ⇒ Carry out alarm clear condition ⇒ Approach reference point
27. RESUMEJOGREFDIGIT	Start continuation of selected program mode (Jog, Reference Point or Digitalize). (VDI signal: NC Start)	1. Jog movement active 2. an alarm response is provided which prevents start or forces deceleration.	⇒ - ⇒ Carry out alarm clear condition
28. STARTDIGITIZE	Start program execution in Digitalize submode (VDI signal: NC Start)	1. Jog movement is active 2. an alarm response is provided which prevents start or forces deceleration. 3. reference-point approach is not yet carried out	⇒ - ⇒ Carry out alarm clear condition ⇒ Approach reference point
29. STOPALL	Stop all axes. (VDI signal: Stop All or Reset key)		
30. STOPPROG	Carry out program stop. (NC block: M0)		
31. STOPJOGREF	Stop JOG movement (VDI signal: NC Stop)		
32. STOPDIGITIZE	Stop digitalization. (VDI signal: NC Stop)		
33. STARTSIG	Start selected machining. (VDI signal: NC Start)	1. Process switch active (mode change, Digitalize ON/OFF, Overstore ON/OFF) 2. an alarm response is provided which prevents start or forces deceleration. 3. a process is running (NC program, block search, loading of machine data)	⇒ - ⇒ Carry out alarm clear condition ⇒ -
34. STOPSIG	Stop active machining. (VDI signal: NC Stop)		
35. INITIALINISTART	Start machine data processing (INI file is already in NCK), (PI command)		
36. INITIALINIEXTSTART	Start machine data processing (INI file is on an external source, e.g. on HMI), (PI command)		
37. BAGSTOP_SLBTYPA	Stop due to single block mode. (VDI signal, individual type A, after stop in other channel of this mode group		
38. BAGSTOPATEND_ SLBTYPB	Stop due to single block mode. VDI signal, individual type B (any blocks), after stop at end of block in other channel of this mode group		

No./Name	Explanation	Not allowed if	Remedy
39. OVERSTORE_ BUFFER_END_ REACHED	Stop because end of overstore buffer "_N_OSTOREXX_SYF" has been reached.		
40. PREP_STOP	Start block search (NC block: Stopre)		
41. PROG_STOP	Stop program execution at end of block (NC block: M00/M01)		
42. STOPPROGABLOCK END	Stop program execution at end of block (alarm, VDI signal: NC stop at end of block)		
43. STOPPROGATASUP END	Stop at ASUP end, if start has been carried out from "Stopped".		
44. PROGSELECT	Select program. (PI command)		
45. PROGSELECTEXT	Select the program which is already on an external source. (PI command)		
46. CHANNEL_PROG SELECT	Select program from a different channel. (channel communication, NC block: INIT)		
47. ASUPDEFINITION	Save definition of ASUP which can be activated. (PI command)		
48. NEWCONF	Sets all machine data with (NEW_CONF) attribute to active state (PI command)		
49. CLEARCANCELALARM	Clear all alarms with the reset condition CANCELCLEAR. (PI command, acknowledge Alarm key)		
50. BLOCKSEARCHUN_ CONTINUE	Continue block search. (NC block: STOPRE		
51. BLOCKSEARCHRUN_ START	Start block search. (PI command)		
52. BLOCKSEARCHRUN_ RESUME	Continue block search. (PI command)		
53. DIGITIZEON	Activate digitizing. (PI command)		
54. DIGITIZEOFF	Deactivate digitizing. (PI command)		
55. FUNCTGENON	Start function generator. (PI command)		
56. FUNCTGENOFF	Switch off the function generator (PI command)		
57. WAITM	Wait for program marker. (channel communication, NC block: WAITM)		
58. WAITE	Wait for program end. (channel communication, NC block: WAITE)		
59. INIT_SYNC	Program selection from another channel with synchronization. (channel communication, NC block: INIT)		
60. HMICMD	Wait until acknowledgment is provided from HMI. (NC block, HMI_CMD)		
61. PROGMODESLASHON	Activate block sip of blocks that can be skipped. (VDI signal: Skip block)	nesting depth too high:	⇒ Wait until the preceding Asup is completed, or abort program

No./Name	Explanation	Not allowed if	Remedy
62. PROGMODESLASH OFF	Deactivate block skip of blocks that can be skipped. (VDI signal: Skip block)	nesting depth too high:	⇒ Wait until the preceding Asup is completed, or abort program
63. PROGMODEDRYRUN ON	Activate test run (VDI signal: Rapid traverse superimposition)	1. nesting depth too high 2. brake reorg error.	⇒ Wait until the preceding Asup is completed, or abort program ⇒ Cancel the program.
64. PROGMODEDRYRUN OFF	Deactivate test run (VDI signal: Rapid traverse superimposition)	1. nesting depth too high: 2. brake reorg error.	⇒ Wait until the preceding Asup is completed, or abort program ⇒ Cancel the program.
65. BLOCKREADINHIBIT_ ON	Activate read-in disable for main run block. (VDI signal: Read-in inhibit)		
66. BLOCKREADINHIBIT_ OFF	Deactivate read-in disable for main run block. (VDI signal: Read-in inhibit)		
67. STOPATEND_ALARM	Stop at end of block. (alarm)		
68. STOP_ALARM	Stop all axes. (alarm)		
69. PROGESTON	Activate program test. (VDI signal: Program test)	1. tool management is active. 2. the NCK channel condition is not READY	⇒ Save tool data ⇒ Press RESET to cancel the program or the process, or wait for the end of the program.
70. PROGTESTOFF	Deactivate program test. (VDI signal: Program test)	the NCK channel condition is not READY	⇒ Press RESET to cancel the program or the process, or wait for the end of the program.
71. STOPATIPOBUFFER_ ISEMPTY_ALARM	Stop at end of block preparation. (alarm)		
72. STOPATIPOBUF_ EMPTY_ALARM_ REORG	Stop at end of block preparation with following reorganization of block execution. (alarm)	nesting depth too high:	⇒ Wait until the preceding Asup is completed, or abort program
73. CONDITIONAL_STOP ATEND	Conditioned stop at block end. (If after continuation by NC Start a reason for stop "Stop at end of block" is still provided, Stop is carried out again.)		
74. CONDITIONAL_SBL_ DEC_STOPATEND	Conditioned stop at block end. (Despite Start, the interpreter or block search does not load a block into the main run)		
75. INTERPRETERSTOP_ ALARM	Stop block search. (alarm)		
76. RETREAT_MOVE_ THREAD	Retraction movement in the case of G33 and stop.		
77. WAITMC	Conditioned waiting for program marker (NC block: WAITMC)		
78. SETM	Set marker. (NC block: SETM)		
79. CLEARM	Delete marker (NC block: CLEARM)		
80. BLOCK_SELECT	Select NC block. (PI command)		
81. LOCK_FOR_EDIT	Disable the currently executed NC program for editing. (PI command)		

No./Name	Explanation	Not allowed if	Remedy
82. START_TEACHINPROG	Start a program in TEACHIN submode. (VDI signal: NC Start)	see 33 and 5	
83. RESUME_TEACHIN PROG	Continue a program in TEACHIN submode. (VDI signal: NC Start)	see 33 and 5	
84. PURE_REORG	Reorganize block execution.		
85. INTERRUPT_ TOPROG_NOREPOS	Activate an ASUP user interrupt in a manual mode. (VDI signal, ASUP, digital-analog interface)	See 10	
86. INTERRUPT_START	Carry out an ASUP user interrupt. Is only carried out in channel state READY. (VDI signal, ASUP, digital-analog interface)	See 10	
87. INTERRUPT_SIGNAL	Carry out a user interrupt of the type "ASUP" (VDI signal, ASUP, digital-analog interface) Group event for all interrupt signals. This event decides on the particular interrupt you wish to trigger. Possible options are: 10, 11, 12, 85, 86.	See 10	
88. STOPBAG	Stop program execution (VDI signal: BAG_Stop)		
89. NEWCONF_PREP_ STOP	Enable all machine data with the attribute (NEW_CONF). (NC_block: NEW_CONF)		
90. BLOCKSEARCHRUN_ NEWCONF	Enable all machine data with the attribute (NEW_CONF). (NC block: NEW_CONF during block search)		
91. CONTINUE_INTERPR	BSALARMEVENTPAR_ CONTINUE_INTERPR Start continuation of interpreter processing (internal block search stop)		
92. SLAVEDATA	Locking for data saving	the NC channel is not stopped.	
93. SET_USER_DATA	Enable the user data, i.e. new tool lengths changed via HMI come into effect in the current program immediately.	1. the NC channel is not stopped. 2. the channel is stopped and the current channel cannot be reorganized.	⇒ Press Stop key/single block/Reset/StopAtEnd key (in Auto). ⇒ Activate block change until the NC block can be reorganized
94. PLCVERSION	Write user PLC version in version file.		
95. CONVERT_SCALING_S YSTEM	BSALARMEVENTPAR_ CONVERT_SCALING_S YSTEM Switch over PI-service unit system		

Glossary / Abbreviations 2

2.1 Abbreviations

A	Output
ASCII	American Standard Code for Information Interchange: American Standard Code for Information Interchange
DB	Data block
DIN	German Industrial Standards
DIO	Data Input/Output: Data transfer display
DRY	Dry Run: Dry run feed
E	Input
EIA code	Special tape code, number of holes per characters always odd
EPROM	Program memory with fixed program
E/R	Controlled Supply and Energy Recovery Module
ETC	ETC key: Extension of the softkey bar in the same menu
FFS	Flash File System
FRAME	Coordinate conversion with the components zero offset, rotation, scaling, mirror-imaging

TRC	Tool radius compensation
GUD	Global User Data: Global User Data
HMI	Human Machine Interface
HW	Hardware
IM	Interface module: Interface Module
IM-S/R	Interface module (S=send/R=receive) Interface module for send/receive mode
INC	Increment: Incremental dimension
ISO code	Special tape code, number of holes per character always even
K1...K4	Channel 1 to Channel 4
K_v	Loop-gain factor
$K_ü$	Transmission ratio
LUD	Local User Data
MB	Megabyte
MD	Machine data
MCS	Machine coordinate system
MDA	Manual Data Automatic: Manual input
MLFB	Machine-readable product designation; order no.
MPF	Main Program File: NC part program (main program)
MPI	Multi Point Interface: Multi Point Interface

MCP	Machine control panel
NC	Numerical Control: CNC; numerical control system
NCK	Numerical Control Kernel: Numerical kernel with block preparation, traversing range etc.
OEM	Original Equipment Manufacturer
OP	Operator Panel: Operator Panel
PC	Personal Computer
PCMCIA	Personal Computer Memory Card International Association: Personal Computer Memory Card International Association:
PG	Programming Device
PLC	Programmable Logic Control
PRT	Program test
RAM	Random Access Memory
RPA	R parameter active: Memory area in NC for R parameter numbers
SBL	Single Block: Single block mode
SBL	Single Block Decoding
SEA	Setting data active: Memory area for setting data in the NC
SD	Setting data
PLC	Programmable Logic Controller
CRC	Cutter Radius Compensation

LEC	Leadscrew error compensation
SSI	Serial Synchronous Interface: Serial Synchronous Interface
SW	Software
TEA	Testing data active: refers to the machine data
TO	Tool offset
TOA	Tool offset active: Memory area for tool offsets
V	Bit type PLC variable

2.2 Glossary

User program	Total of all status bar graphs in an executable form in the PLC
Command	Statement in the user program
Operating mode	Mode of *program execution*, e.g. manual mode, automatic mode, corresponding to the machine operation
Diagnosis	Detection of faulty processes during *machining*; displays undesired or unexpected phenomena during the process
Multi-point interface	MPI Hardware module for online coupling with the user program
Sensor	Electric *element*; provides a *signal* to the *control system*

To
SIEMENS AG

A&D MC IS
Postfach 3180

D-91050 Erlangen, Germany

(Telephone: +49 (0) 180 / 538 – 8008 [hot-line]
Telefax: +49 (0) 9131 / 98 – 1145
Mailto: motioncontrol.docu@erlf.siemens.de)

Suggestions

Corrections

For publication/manual:

SINUMERIK 802D

User Documentation

From

Name

Company/Dept.

Address

Zip code:City:

Telephone:

Telefax:

Diagnostics Guide

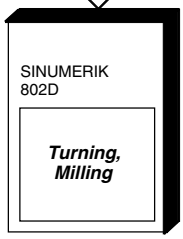
Order No.: 6FC5698-2AA20-0BP1
Edition: 07.02

Should you come across any printing errors when reading this publication, please notify us on this sheet.Suggestions for improvement are also welcome.

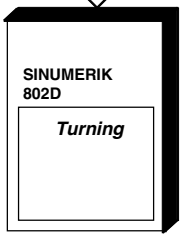
Suggestions and/or corrections

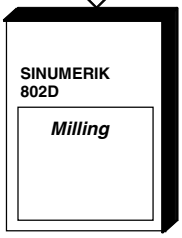
Document Structure SINUMERIK 802D

General Documentation: **Catalog**

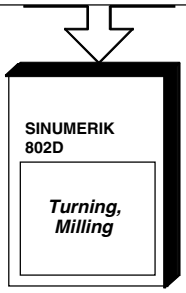


User's Guide: **Operation and Programming**

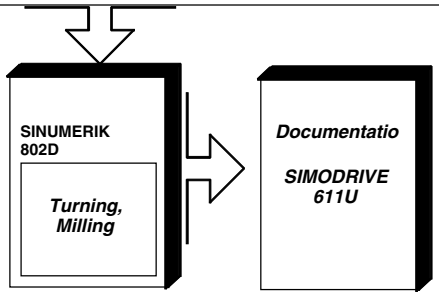




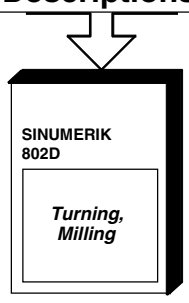
User's Guide: **Diagnostics Guide**



Technical Manual: **Start-up**



Technical Manual: **Descriptions of Functions**



Siemens AG
Automation and Drives
Motion Control Systems
Postfach 3180, D-91050 Erlangen
Germany
www.ad.siemens.de

Order No. 6FC5698-2AA20-0BP1