

# sinumerik

SINUMERIK 802S base line  
SINUMERIK 802C base line

**SIEMENS**



# SIEMENS

## SINUMERIK 802S base line SINUMERIK 802C base line

### Diagnostics Guide

### User Documentation

Alarms	1
--------	---

Abbreviations/Glossary	2
------------------------	---

### Valid as from

#### *Control system*

SINUMERIK 802S base line

SINUMERIK 802C base line

#### *Software version*

4

4

# SINUMERIK<sup>®</sup> Documentation

## Key to editions

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

The column headed "Note" lists the amended sections, with reference to the previous edition.

Marking of edition in the "Note" column:

- A** ... .. New documentation.
- B** ... .. Unchanged reprint with new order number.
- C** ... .. Revised edition of new issue.

<b>Edition</b>	<b>Order No.</b>	<b>Note</b>
1999.02	6FC5598-2AA20-0BP1	<b>A</b>
2000.04	6FC5598-2AA20-0BP1	<b>A</b>
2002.01	6FC5598-3AA20-0BP2	<b>C</b>
2003.08	6FC5598-4AA21-0BP0	<b>A</b>

## Trademarks

SIMATIC<sup>®</sup>, SIMATIC HMI<sup>®</sup>, SIMATIC NET<sup>®</sup>, SIMODRIVE<sup>®</sup>, SINUMERIK<sup>®</sup>, and SIMOTION<sup>®</sup> are registered trademarks of SIEMENS AG.

Other names in this publication might be trademarks whose use by a third party for his own purposes may violate the registered holder.

### Copyright Siemens AG 2003. All right reserved

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model, are reserved.

### Exclusion of liability

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

© Siemens AG, 2003  
Subject to technical changes without notice.

## Preface

The present description is intended as a manual for the machine tool operator so that he can:

- assess special events when operating the machine correctly
- learn the response of the equipment on special events
- use the provided possibilities to continue work after the special event
- follow the notes referring him to documentation with further information.

### Scope

This Description includes the alarms that may occur in the following areas: NC kernel (NCK), cycles and PLC.

Further alarms provided from the MMC area (Man Machine Communication) can also occur. They are announced to the user in self-explaining alarm lines on the operator panel. They are not subject of the Diagnostic Instructions. For special cases in conjunction with the integrated PLC refer to literature on the SIMATIC S7-300 system.

### Sorting

The alarms are sorted in the Diagnostic Instructions by ascending alarm numbers with spaces between them.

### Safety



#### Danger

Please check the situation of the equipment carefully using the description of the particular alarm occurred. Eliminate the causes of occurrence of the alarm and acknowledge it as specified. Inattention could cause damage to machine, workpiece, stored settings and - under certain circumstances - to your health.

### NCK alarms

Table 1-1 Alarm number ranges

000 000 - 009 999	General alarms	
010 000 - 019 999	Channel alarms	
020 000 - 029 999	Axis/spindle alarms	
030 000 - 099 999	Functional alarms	
060 000 - 064 999	SIEMENS cycle alarms	
065 000 - 069 999	User cycle alarms	
070 000 - 079 999	Manufacturer and OEM compile cycles	

**MMC alarms/messages**

Table 1-2 Alarm number ranges (continued)

100 000 - 100 999	Basic system	MMC0
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

**611D alarms**

Table 1-3 Alarm number ranges (continued)

300 000 - 399 999		
-------------------	--	--

**PLC alarms/messages**

Table 1-4 Alarm number ranges (continued)

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

# Contents

<b>1. Alarms</b>	1-1
1.1 Overview of NCK Alarms	1-2
1.2 Cycle Alarms	1-82
1.3 PLC Alarms	1-88
1.4 Action List	1-91
<b>2. Abbreviations/Glossary</b>	2-1
2.1 Abbreviations	2-1
2.2 Glossary	2-5





# Alarms

# 1

## System errors

The alarms listed in the following are system errors indicating internal error conditions. The internal error number displayed together with the error message provides the developer with important information on error cause and error location.

These system errors will not be described in detail. As far as they occur on the delivered control systems at all, please contact the following hotline, specifying alarm number, alarm text and internal system error number contained in the error message:

### Hotline Germany

Siemens AG, A&D techsupport

Telephone: 0180 50 50-222

Mailto: [techsupport@ad.siemens.de](mailto:techsupport@ad.siemens.de)

### Helpline

Telephone: 0049 180 50 50 111

### Hot line China

SLC A&D Customer Support

Telephone: 010-64719990

Fax: 010-64719991

## 1.1 Overview of NCK Alarms

<b>1 002</b>	System error %1
<b>Explanation</b>	%1 = System error 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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>1 003</b>	Alarm pointer for this self-clearing alarm %1 is zero
<b>Explanation</b>	%1 = Alarm number  The address used by the operating system for self-clearing alarms (zero pointer) is not permitted for use in the system.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>1 004</b>	Alarm reaction from NCK alarm is false projected
<b>Explanation</b>	%1 = incorrect alarm number  The alarm response configured by the operating system is incorrect.
<b>Response</b>	Alarm display Interface signals are set NC not ready for operation.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Power ON
<b>1 005</b>	Operating system error %1
<b>Explanation</b>	%1 = Operating system error  This alarm indicates that the operating system has detected a severe error in the system.
<b>Response</b>	Alarm display. Interface signals are set. NC not ready for operation. NC Start inhibited. NC Stop at alarm.

<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Power ON
<b>1 010</b>	Channel %1 system error %2
<b>Explanation</b>	%1 = Channel number %2 = System error 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.
<b>Response</b>	Alarm display. Interface signals are set. NC not ready for operation. NC Start inhibited. NC Stop at alarm.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Power ON
<b>1 011</b>	Channel %1 %3 system error %2
<b>Explanation</b>	%1 = Channel number %2 = System error number %3 = Channel parameter: Block number, label 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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>1 012</b>	Channel %1 system error %2
<b>Explanation</b>	%1 = Channel number %2 = System error 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.
<b>Response</b>	Alarm display.

**Remedy** Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**1 014** Channel %1 system error %2

**Explanation** %1 = Channel number  
%2 = System error 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.

**Response** Alarm display.  
Interface signals are set.  
NC Stop at alarm.  
NC Start inhibited.

**Remedy** Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**1 015** Channel %1 axis %2 system error %3

**Explanation** %1 = Channel number  
%2 = Axis number  
%3 = System error 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.

**Response** Alarm display.  
Interface signals are set.  
Channel not ready for operation.

**Remedy** Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**1 016** Channel %1 axis %2 system error %3

**Explanation** %1 = Channel number  
%2 = Axis number  
%3 = System error 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.

**Response** Alarm display.  
Interface signals are set.

<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>1 017</b>	Channel %1 axis %2 system error %3
<b>Explanation</b>	%1 = Channel number %2 = Axis number %3 = System error 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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>1 018</b>	Floating point arithmetic error in channel %1 task %2 station %3 FPU state: %4
<b>Explanation</b>	%1 = Channel number %2 = Task ID %3 = Station priority %4 = FPU state  The floating point arithmetic unit of the processor has detected an error.
<b>Response</b>	Alarm display. Interface signals are set. Channel not ready for operation. NC Stop at alarm. NC Start inhibited.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>1 019</b>	Floating point arithmetic error at address %3 in channel %1 task %2 FPU state %4
<b>Explanation</b>	%1 = Channel number %2 = Task ID %3 = Code address of the operation which has caused the error %4 = FPU state  Due to a calculation error, the floating point arithmetic unit of the processor has initiated an exception.

<b>Response</b>	Alarm display. Interface signals are set. Channel not ready for operation. NC Stop at alarm. NC Start inhibited.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>2 000</b>	Sign of life monitoring: PLC not alive
<b>Explanation</b>	The PLC must provide a sign of life within a certain period of time. If not, this alarm is output. This alarm is also output as the consequence of a PLC stop (PLC stop with programming tool, PLC stop from start-up switch, PLC stop from alarm).
<b>Response</b>	NC Start inhibited. NC not ready for operation. Mode group ('BAG') not ready for operation. NC Stop at alarm. Alarm display. Interface signals are set.
<b>Remedy</b>	If none of the cases mentioned above exists, please contact the hotline indicated in the beginning of this Manual and specify the operating system error number.
<b>Program continuation by</b>	Power ON
<b>2 001</b>	PLC has not started up
<b>Explanation</b>	The PLC must provide at least one sign of life within the defined period of time after Power On.
<b>Response</b>	NC Start inhibited. NC not ready for operation. Mode group ('BAG') not ready for operation. NC Stop at alarm. Alarm display. Interface signals are set.
<b>Remedy</b>	Please contact the hotline indicated in the beginning of this Manual.
<b>Program continuation by</b>	Power ON

<b>2 140</b>	The actual service switch position forces the SRAM to be cleared at the next Power on (general reset active)
<b>Explanation</b>	The initialization switch is now set to General Reset. This forces the SRAM of the module to be cleared when the module is reset next time. As a consequence, the NC data memory gets lost.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Reset initialization switch to "1".
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>3 000</b>	Emergency Stop
<b>Explanation</b>	EMERGENCY STOP request is provided at the NCK/PLC interface (V 26000000.1).
<b>Response</b>	NC Start inhibited. NC Stop at alarm Alarm display. Interface signals are set.
<b>Remedy</b>	Check whether an EMERGENCY STOP cam has been approached or an EMERGENCY STOP button has been actuated. Check the PLC user program. Eliminate the EMERGENCY STOP cause and acknowledge EMERGENCY STOP via the PLC /NCK interface (V 26000000.2).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>3 001</b>	Internal emergency stop
<b>Explanation</b>	This alarm is not displayed.
<b>Response</b>	NC Start inhibited. NC Stop at alarm.
<b>Remedy Z</b>	No remedial action necessary.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program
<b>4 060</b>	Standard machine data loaded
<b>Explanation</b>	Booting with standard values by: <ul style="list-style-type: none"> <li>• Operating action (e.g. start-up switch)</li> <li>• MD 11200 INIT_MD</li> <li>• Loss of retentive data</li> <li>• Operating action "Booting with saved data", without data saving beforehand</li> </ul>
<b>Response</b>	Alarm display
<b>Remedy</b>	The specific data of the equipment on hand must be entered/loaded after the standard machine data (MD) have been loaded automatically.

**Program continuation by** Press the Clear key to clear the alarm. Reload your own machine data.

**4 062**

Backup data loaded

**Explanation**

The user data saved to FLASH have been loaded into the SRAM.

**Response**

Alarm display

**Remedy**

Reload your own machine data.

**Program continuation by**

Press the RESET key to clear the alarm.

**4 065**

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

**Explanation**

During power-up, it has been detected that inconsistencies are possible in the battery-backed memory.

The battery-backed memory was initialized using the last back-up copy. As a result, the changes in the battery-backed memory, which have been made since the last update of the backup copy, were lost. This may be caused by the fact that the buffer time has been exceeded. Please make sure that the appropriate on-time of your control system complies with the on-time required by the Start-up Guide.

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.

**Response**

Alarm display

Interface signals are set.

NC Start inhibited

**Remedy**

Restart the control system.

**Program continuation by**

**4 075**

Machine data %1 (and may be others) not altered - permission level %2 needed

**Explanation**

%1 = String: MD identifier

%2 = Write protection level of the MD

When executing a TOA file, you have tried to write into data whose protection level is higher than the right to access currently set on the control. Your value entry will be denied.

This alarm is only set when the first violation of the right to access is detected.

**Response**

Alarm display.

**Remedy**

Enter the password to set the required access level or delete the respective machine data from the MD file.

**Program continuation by**

Press the Clear key to clear the alarm. No further operation required.



<b>4 076</b>	%1 machine data could not be modified with permission level %2
<b>Explanation</b>	<p>%1 = Number of MD</p> <p>%2 = Set user class</p> <p>When executing a TOA file, you have tried to write into data whose protection level is higher than the right to access currently set on the control. Your value entry will be denied.</p> <p>This alarm is output when alarm 4075 is acknowledged. It can only be cleared by Power On.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Set the required access level either using the key-operated switch or be entering the password and/or delete the respective machine data from the MD file.
<b>Program continuation by</b>	Power ON
<b>4 111</b>	PLC cycle increased to %1 ms
<b>Explanation</b>	<p>The PLC cycle divider has not been set to an integer multiple of the IPO cycle divider.</p> <p>The divider (MD 10 074 PLC_IPO_TIME_RATIO) has been increased.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Adapt machine data.
<b>Program continuation by</b>	Power ON
<b>4 230</b>	Channel %1 data alteration from external not possible in current channel state
<b>Explanation</b>	<p>%1 = Channel number</p> <p>These data may not be entered during part program execution (e.g. setting data for spindle speed limitation or for dry run feed).</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	The data to be entered must be altered before starting the part program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>4 310</b>	Declaration in machine data %1 index %2 is not allowed
<b>Explanation</b>	<p>%1 = String: MD identifier</p> <p>%2 = Index in MD array</p> <p>The values of the machine data in the array must be arranged in ascending order.</p>
<b>Response</b>	<p>NC not ready for operation.</p> <p>NC Stop at alarm.</p> <p>NC Start inhibited.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	Correct MD

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>4 400</b>	Machine data alteration will cause reorganization of buffered memory (loss of data!)
<b>Explanation</b>	An MD has been altered which configures the buffered memory. An NCK boot with the altered data will reorganize the buffered memory and thus result in a loss of all buffered user data (part programs, tool data, GUD, LEC, ..).
<b>Response</b>	Alarm display.
<b>Remedy</b>	If the control system contains user data which have not yet been backed up, these data must be saved before the next NCK boot. The reorganization of the memory can be avoided by resetting the altered MD to the value during the last boot manually.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>5 000</b>	Communication job not executed
<b>Explanation</b>	The communication task (data exchange between NCK and MMC), e.g. loading of an NC part program) cannot be executed due to insufficient memory. Cause: too many parallel communication jobs.
<b>Response</b>	Alarm display.
<b>Remedy</b>	No remedial action possible - the operating action that resulted in the alarm message must be repeated. Press Cancel to clear the alarm display.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>6 000</b>	Memory reorganized using standard machine data
<b>Explanation</b>	The memory management could not realize the NC user memory mapping with the values provided in the machine data, because the total memory is used by the NC user both as a dynamic and a static memory (e.g. for the number of tool offsets, number of directories and files, etc.) and is therefore not sufficient.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited. NC Stop at alarm.
<b>Remedy</b>	Redefine NC memory mapping! It is not possible to specify a certain MD as an alarm cause for the NC user memory assignment. The MD that has caused the alarm must therefore be determined on the base of the default values in the machine data and by modifying the user-specific memory assignment 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 by a certain fraction.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

---

<b>6 020</b>	Machine data have been altered - now memory is reorganized
<b>Explanation</b>	Machine data defining the NC user memory mapping have been altered. The data management has re-mapped the memory with respect of the altered machine data.
<b>Response</b>	Alarm display.
<b>Remedy</b>	No remedial action required. Any user data needed must be re-entered.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>6 500</b>	NC memory is full
<b>Explanation</b>	Too many part programs have been loaded. The job cannot be carried out. This error may concern files of the NC file system (part of NC memory), such as initialization files, NC programs, etc., during commissioning.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Delete or unload files (e.g. part programs).
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>6 510</b>	Too many files in the NC memory
<b>Explanation</b>	The number of files in the NC file system (part of NC memory) has reached the maximum.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Delete or unload files (e.g. part programs).
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>6 530</b>	Too many files in directory
<b>Explanation</b>	The number of files in a directory of the NC memory has reached the maximum.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Delete or unload files (e.g. part programs) in the directory concerned.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>6 540</b>	Too many directories in the NC memory
<b>Explanation</b>	The number of directories in the NC file system (part of NC memory) has reached the maximum.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Delete or unload directories not needed (e.g. workpiece).
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>6 560</b>	Data format not allowed
<b>Explanation</b>	Invalid data have been entered in a file of the NC, e.g. binary data are loaded into the NC as an ASCII file.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Mark the file as binary data (e.g. extension: .BIN)
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>6 570</b>	NC memory is full
<b>Explanation</b>	The DRAM file system of the NCK is full. The job cannot be executed. Too many system files have been created in the DRAM.
<b>Response</b>	Alarm display
<b>Remedy</b>	Start less "Process from External" operations.
<b>Program continuation by</b>	Press the Clear key to clear the alarm.
<b>8 040</b>	Machine data %1 reset, corresponding option is not set
<b>Explanation</b>	%1 = String: MD identifier An MD has been set which is locked by an option.
<b>Response</b>	Alarm display.
<b>Remedy</b>	To upgrade your CNC with the required option, contact either your machine manufacturer or a sales engineer of AUT 2 of SIEMENS AG.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>10 203</b>	Channel %1 NC-start not possible with unreferenced axes (Cmd = %2).
<b>Explanation</b>	%1 = Channel number %2 = String (event name) 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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Referencing start can be initiated either channel or axis-specifically. <ul style="list-style-type: none"> <li>1. Channel-specific reference-point approach: The rising edge of the interface signal "Activate referencing" (V 32000001.0) starts an automatic cycle which start the axes of the channel in the sequence of order specified in the axis-specific MD 34 110 REFP_CYCLE_NR (axis order for channel-specific referencing). <ul style="list-style-type: none"> <li>-1: Axis is not involved in channel-specific referencing, but must be referenced for NC start.</li> <li>0: Axis is not involved in channel-specific referencing, but must be referenced for NC start</li> <li>1-4: Starting sequence for channel-specific referencing (simultaneous start with the same number.)</li> </ul> </li> </ul>

	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 Minus Direction).
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>10 208</b>	Channel %1 Continue program with NC-Start
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. NC Stop at alarm.
<b>Remedy</b>	Press NC Start.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>10 225</b>	Channel %1: command % 2 refused
<b>Explanation</b>	%1 = Channel number %2 = String (event name) The channel contained a command which cannot be executed.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Press RESET.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>10 600</b>	Channel %1 block %2 auxiliary function during thread cutting active
<b>Explanation</b>	%1 = Channel number %2 = Block number, label An auxiliary function output is programmed in a thread block.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Subsequent error might occur if the machining path of the thread block is too short and further blocks (thread blocks) follow in which no machining stop may occur. Possible remedial actions: <ul style="list-style-type: none"> <li>• Program longer path and/or reduce traversing speed.</li> <li>• Output auxiliary function in another block (program section).</li> </ul>
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>10 601</b>	Channel %1 block %2 zero-velocity at end point during thread cutting
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>This alarm occurs only if several blocks with G33 follow one after another. The block end velocity in the specified block is zero although still one more velocity block follows. Possible causes are, for example:</p> <p>G09                  Auxiliary function after movement                  Output of auxiliary function prior to the movement of the subsequent block</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Modify NC part program (do not program "Stop at block end" G09)
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 602</b>	Channel %1 block %2 velocity limitation during thread cutting
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>The axis in the displayed thread block would exceed its maximum velocity when the spindle override is in the maximum position.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	If the axis velocity is not limited (error-free thread), no remedial action is required. Otherwise, the spindle speed for the thread block must be reduced.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>10 620</b>	Channel %1 block %3 axis %2 at software limit switch %4
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name, spindle number                  %3 = Block number, label                  %4 = String</p> <p>During the traversing movement, it is detected that the software limit switch has been overtraveled in the displayed direction. Overtraveling the traversing range could not be detected during block preparation (e.g. circular interpolation), or a coordinate transformation is active.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  NC Start inhibited.</p>
<b>Remedy</b>	Check set and programmed zero offsets.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>10 621</b>	Channel %1 axis %2 rests on software limit switch %3
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number %3 = String The axis specified is already on the displayed software limit position.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Machine data: Check the machine data POS_LIMIT_PLUS/POS_LIMIT_PLUS2 (MD 36 110/MD 36 130) and POS_LIMIT_MINUS/POS_LIMIT_MINUS2 (MD 36 100/MD 36 120) 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); make sure that the 2nd software limit switch is selected.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.
<b>10 631</b>	Channel %1 axis %2 rests on software limit switch %3
<b>Explanation</b>	%1 = Channel number %2 = Axis, spindle %3 = String (+ or -) IN the JOG mode, the specified axis reaches the work area limitation (positive at "+", negative at "-").
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check the work area limitation setting data 43420 WORK AREA_LIMIT_PLUS and 43430 WORK AREA_LIMIT_MINUS.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.
<b>10 640</b>	Channel %1 block %3 spindle %2 cannot stop during gear change
<b>Explanation</b>	%1 = Channel number %2 = Spindle number %3 = Block number, label The spindle is in oscillation mode for gear stage change and waits for PLC checkback that gear stage change has been carried out (interface signal: Gear changed V 38032000.3). During this time, no spindle stop by Reset is possible (V 30000000.7) or NC Stop axes plus spindle (V 32000007.4).
<b>Response</b>	Alarm display.
<b>Remedy</b>	No remedial action required. The spindle-specific interface signal Spindle Reset (V 3803000.2) cancels oscillation mode V 38030002.2.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.

<b>10 720</b>	Channel %1 block %3 axis %2 software limit switch %4
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name, spindle number                  %3 = Block number, label                  %4 = String (+ or -)</p> <p>The path programmed for the axis violates the currently active software limit switch. (The 2nd software limit switch becomes active with the interface signal "2nd software limit switch plus/minus" in V 380x1000.2 and .3). The alarm is activated when preparing the part program block.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  NC Start inhibited.</p>
<b>Remedy</b>	<p>Check the position specified for the axis in the part program.                  Check the machine data                  POS_LIMIT_MINUS/POS_LIMIT_MINUS2 (MD 36 110/MD 36 130) and                  POS_LIMIT_PLUS/POS_LIMIT_PLUS2 (MD 36 100/MD36 120)                  for the software limit switches.                  Check the axis-specific interface signals: "2nd software limit switch plus/minus"                  (V 380x1000.2 and .3) and make sure that the 2nd software limit switch is                  selected.                  Check the currently active zero offsets via the current frame.</p>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 730</b>	Channel %1 block %3 axis %2 work area limitation %4
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name, spindle number                  %3 = Block number, label                  %4 = String (+ or -)</p> <p>During block preparation, it has been detected that the programmed path carries the axes across the work area limitation.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Include correction block in reorganization.                  NC Start inhibited.</p>
<b>Remedy</b>	<p>Modify the part program.                  Change the work area limitation in the setting data.</p>
<b>Program continuation by</b>	Press the N C START to clear the alarm and to continue the program.
<b>10 750</b>	Channel %1 block %2 tool radius compensation activated without tool no.
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>A tool T.. must be selected so that the control system can consider the associated compensation values.</p>



	<p>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).</p> <p>The tool radius compensation (TRC) will be included in calculation if the function G41 or G42 is programmed. The compensation values are contained in the parameter P6 (geometry value) and P15 (wear value) of the active compensation data block D<sub>x</sub>.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Program a tool No. under address T.. with G41/G42 before calling the tool radius compensation.</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>10 751</b>	<p>Channel %1 block %2 danger of collision due to tool radius compensation</p>
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>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.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>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 elongated or intermediate blocks are inserted so that an intersection point is always provided).</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>10 752</b>	<p>Channel %1 block %2 overflow of local block buffer with tool radius compensation</p>
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>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.</p> <p>The size of the buffer memory is fixed by the system and cannot be altered via MD.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>

<b>Remedy</b>	Reduction of the buffer memory which has been occupied by modifying the NC program. This can be achieved if you avoid the following: <ul style="list-style-type: none"> <li>• blocks without traversing information in the compensation level</li> <li>• 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).</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 753</b>	Channel %1 block %2 activate tool radius compensation in linear block only
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  Tool radius compensation with G41/G42 may only be selected on blocks with the G function G00 (rapid traverse) or G01 (feed) active.  The block with G41/G42 must contain at least one axis of the level G17 to G19; always both axes are recommended, since both axes are traversed when selecting the compensation.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	Correct the NC program; replace the compensation selection to a block with linear interpolation.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 754</b>	Channel %1 block %2 deactivate tool radius compensation in linear block only
<b>Explanation</b>	%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.  The block with G40 must contain at least one axis of the plane G17 to G19; both axes are always recommended, since usually both axes traverse when deselecting the compensation.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	Correct NC program; replace compensation deselection to a block with linear interpolation.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>10 755</b>	Channel %1 block %2 do not activate tool radius compensation via KONT at the current starting point
<b>Explanation</b>	<p>%1 = Channel number %2 = Block number, label</p> <p>When activating the cutter radius compensation via KONT, the start point of the approach block is within the compensation circle, thus already violating the contour.</p> <p>When deselecting the cutter radius compensation with G41/G42, the approach behavior (NORM or KONT) determines the movement along the contour if the current actual position is behind the contour. When KONT is active, a circle with the cutter radius is drawn around the programmed start point (= end point of approach block). The tangent that leads through the current actual position and that does not violate the contour is the approach movement.</p> <p>If the start point is within the compensation circle and around the target point, the tangent will lead through this point.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Place the selection of the cutter radius compensation such that the start point of the approach movement will be outside the compensation circle around the target point (programmed traversing movement &gt; compensation radius). The following possibilities are provided:</p> <ul style="list-style-type: none"> <li>• Selection in the preceding block</li> <li>• Insert intermediate block</li> <li>• Select approach behavior NORM</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 756</b>	Channel %1 block %2 do not activate tool radius compensation via KONT at the current starting point
<b>Explanation</b>	<p>%1 = Channel number %2 = Block number, label</p> <p>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.</p> <p>When the cutter radius compensation is deselected with G40, the starting behavior (NORM or KONT) will determine the movement along the contour if the programmed end point is behind the contour. When 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.</p> <p>If the programmed end point is within the compensation circle around the target point, no tangent will lead through this point.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>

<b>Remedy</b>	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: <ul style="list-style-type: none"> <li>• Deselection in the next block</li> <li>• Insert intermediate block</li> <li>• Select starting behavior NORM</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 762</b>	Channel %1 block %2 too many empty blocks between two traversing blocks with active tool radius compensation
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  The maximum admissible number of empty blocks is limited
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	1. Modify 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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 763</b>	Channel %1 block %2 path component of the block in the compensation plane becomes zero.
<b>Explanation</b>	%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 level becomes zero. If no movement information perpendicular to the compensation level is contained in the original block, the block is skipped.
<b>Response</b>	Alarm display.
<b>Remedy</b>	<ul style="list-style-type: none"> <li>• The behavior at narrow places which cannot be machined with the active tool is correct.</li> <li>• Modify part program</li> <li>• If necessary use a tool with smaller radius</li> </ul>
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>10 764</b>	Channel %1 block %2 discontinuous path with active tool radius compensation
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>This alarm occurs when tool radius compensation is active and the start point used for the calculation of the compensation is not equal to the end point of the preceding block.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	Modify part program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 777</b>	Channel %1 block %2 tool radius compensation: too many blocks with suppression of compensation
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>The maximum permissible number of blocks with active compensation suppression when tool radius compensation is active is limited.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<ul style="list-style-type: none"> <li>• Modify part program.</li> <li>• Check whether SBL2 is selected. When SBL2 is active, a block from each part program line is generated what results in that the permissible number of empty blocks between two traversing blocks can be exceeded.</li> </ul>
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>10 778</b>	Channel %1 block %2 Preparation stop with active tool radius compensation
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>When feed stop is detected with tool radius compensation active (either programmed by the user or generated internally), this alarm is set, 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.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	<ul style="list-style-type: none"> <li>• Continue with CANCEL and Start.</li> <li>• Modify part program.</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>10 810</b>	Channel %1 block %2 No master spindle defined
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>A revolutional feedrate has been defined although no master spindle is defined. The spindle rotation is the prerequisite to calculate the revolutional feedrate.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Include the correction block into the reorganization.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	Define a spindle in the machine data.
<b>Program continuation by</b>	Press the NC START to clear the alarm and to continue the program.
<b>10 860</b>	Channel %1 block %2 feed rate not programmed
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>An interpolation type other than G00 (rapid traverse) is active in the displayed block. No F value programmed.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Program the feed value with respect of the interpolation type used.</p> <ul style="list-style-type: none"> <li>• G94 and G97: The feed is programmed under address F in [mm/min] or [m/min].</li> <li>• G95: The feed is programmed as a revolutional feedrate in [mm/rev] under address F.</li> <li>• G96: The feed is programmed as a cutting speed under address S in [m/min]. It results from the current spindle speed.</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 870</b>	Channel %1 block %2 facing axis not defined
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>When the constant cutting speed is selected using the function G96, the spindle speed is controlled via the position of the facing axis such that the cutting speed programmed under S [mm/min] results at the tool edge.</p> <p>In the channel-specific MD DIAMETER_AX_DEF[n,m]=x (n .. channel index, m .. spindle index, x .. axis name), the name of the facing axis [string] can be specified for each of the 5 spindles which is used for speed calculation.</p> $S [1/min] = \frac{S_{G96} [m / min] \cdot 1000}{D_{\text{facing axis}} [mm] \cdot \pi}$

---

<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	Enter the name of the facing axis in the channel-specific machine data DIAMETER_AX_DEF for the spindles used.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 880</b>	Channel %1 block %2 too many empty blocks between two traversing blocks when inserting chamfer or radius
<b>Explanation</b>	%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).
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	Modify part program such that the permissible number of empty blocks is not exceeded.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 881</b>	Channel %1 block %2 overflow of local block buffer when inserting chamfer or radius
<b>Explanation</b>	%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..
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited.
<b>Remedy</b>	Modify the part program such that the number of empty blocks is reduced.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 882</b>	Channel %1 block %2 do not activate chamfer or radius in blocks without traversing
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  No chamfer or radius has been inserted between 2 linear or circle contours (chamfer edging), because: <ul style="list-style-type: none"> <li>• no straight line or circle contour exists in the plane</li> </ul>

- a movement exists outside the plane
- the level has been changed
- the permissible number of empty blocks without traversing information (dummy blocks) has been exceeded

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

**Remedy** Correct the part program with respect of the error mentioned above.  
**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**10 900** Channel %1 block %2 no S value programmed for constant cutting speed

**Explanation** %1 = Channel number  
%2 = Block number, label

When G96 is active, the constant cutting speed under address S is missing.

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

**Remedy** Program the constant cutting speed under S in [m/min] or deselect function G96. For example, the preceding feed remains when G97 is active but the spindle goes on rotating with the current speed.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**10 910** Channel %1 block %2 excessive velocity of one path axis

**Explanation** %1 = Channel number  
%2 = Block number, label

When transformation is selected, an extreme velocity rise in one or several axes occurs, for example, since the path runs in the vicinity of the pole.

**Response** Alarm display.

**Remedy** Division of the NC block into several blocks (e.g. 3) so that the path section with the chamfer is as small as possible and therefore short. The remaining blocks will then be traversed with the programmed velocity.

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**10 911** Channel %1 block %2 transformation prohibits to traverse the pole.

**Explanation** %1 = Channel number  
%2 = Block number, label

The given curve leads through the pole of the transformation.

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



<b>Remedy</b>	Modify part program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 930</b>	Channel %1 block %2 interpolation type not allowed in stock removal contour
<b>Explanation</b>	<p>%1 = Channel number          %2 = Block number, label</p> <p>The contour of the stock removal cycle (LCYC 95) contains path commands other than G00, G01, G02 or G03. The contour program may only contain contour elements built from these G functions (i.e. no thread blocks, no G05, etc.).</p>
<b>Response</b>	<p>Alarm display.          Interface signals are set.          NC Start inhibited.</p>
<b>Remedy</b>	Program only path elements in the contour program, which consist of straight lines and circular arcs.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 931</b>	Channel %1 block %2 error in programmed stock removal contour
<b>Explanation</b>	<p>%1 = Channel number          %2 = Block number, label</p> <p>The following errors are contained in the stock removal program (LCYC 95) for the contour:</p> <ul style="list-style-type: none"> <li>• full circle</li> <li>• intersecting contour elements</li> <li>• wrong start position</li> </ul>
<b>Response</b>	<p>Alarm display.          Interface signals are set.          NC Start inhibited.</p>
<b>Remedy</b>	Correct the errors mentioned above in the program for the contour to be machined.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>10 932</b>	Channel %1 block %2 preparation of contour has been restarted
<b>Explanation</b>	<p>%1 = Channel number          %2 = Block number, label</p> <p>The stock removal cycle LCYC 95 has been interrupted during the preparation phase of the stock removal contour.</p>
<b>Response</b>	<p>Alarm display.          Interface signals are set.          NC Start inhibited.</p>

**Remedy** Do not admit interruption during the contour preparation in the stock removal cycle LCYC 95.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**10 933** 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 movements in both axes in the machining plane. The stock removal cycle (LCYC 95) has been canceled.

**Response** 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 movements in both axes of the current machining plane.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**10 934** Channel %1 block %2 array for contour segmentation is 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 movements in both axes in the machining plane (LCYC 95).

**Response** 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** Press the RESET key to clear the alarm. Restart part program.

**12 000** 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 search information remains unambiguously.

(e.g. X.. T.. F.. etc. - Exception: G, M functions).

**Response** Alarm display.  
Interface signals are set.

**Remedy** Correct block.

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

<b>12 010</b>	Channel %1 block %2 address %3 address type programmed too often				
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>%3 = Source string of address</p> <p>It is defined for each address type how often it may occur in an NC block (for example, all axes together form a address type, which is also subject to a block limit).</p>				
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p>				
<b>Remedy</b>	Correct the block.				
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.				
<b>12 050</b>	Channel %1 block %2 DIN-address %3 is not configured				
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>%3 = NC address in source text block</p> <p>The name of the NC address (e.g. X, U, X1) is not defined in the control system.</p>				
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Corrected block.</p>				
<b>Remedy</b>	Study Programming Instructions and machine data with refer to the really configured addresses and their meaning and correct the NC block accordingly.				
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.				
<b>12 060</b>	Channel %1 block %2 same G group programmed repeatedly				
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>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.</p> <p>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).</p> <p>G FUNCTIONS:</p> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding: 2px;">Syntax-defining G functions</td> <td style="padding: 2px;">Non-syntax defining G functions</td> </tr> <tr> <td style="padding: 2px;">1st to 4th G group</td> <td style="padding: 2px;">5th to nth G group</td> </tr> </table>	Syntax-defining G functions	Non-syntax defining G functions	1st to 4th G group	5th to nth G group
Syntax-defining G functions	Non-syntax defining G functions				
1st to 4th G group	5th to nth G group				
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p>				
<b>Remedy</b>	No remedial action necessary. However, it should be checked whether the last programmed G function is really the desired one.				

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

**12 070** 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 to 4th G group are syntax-defining.

**Response** Alarm display.

Interface signals are set.

Corrected block.

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

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

**12 080** 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 condition for the jump is missing!

Example 2:

N10 R-50 =12 ; Faulty arithmetic parameter number

**Response** Alarm display.

Interface signals are set.

Corrected block.

**Remedy** Analyze block and correct it using the Programming Instructions.

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

**12 110** Channel %1 block %2 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; No interpolation parameter may be programmed in the linear block.

**Response** Alarm display.

	Interface signals are set.
<b>Remedy</b>	Check block structure and correct it as required by the program.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 120</b>	Channel %1 block %2 Write special G function in separate block
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The G function programmed in this block must be written in a separate block. No general addresses are allowed in the same block. These G functions are: G25, G26      Spindle speed limitation Example: G4 F1000 M100; No M function is allowed in the G4 block.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Program the G function in a separate block.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 140</b>	Channel %1 block %2 expression %3 not contained in this release
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Remove the displayed function from the program.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 150</b>	Channel %1 block %2 operation %3 not compatible with data type
<b>Explanation</b>	%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).
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Modify the definition of the variables used such that the desired operations can be carried out.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.

<b>12 160</b>	Channel %1 block %2 range of values exceeded
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>The constant programmed for the variable exceeds the value range which has been defined by the data type in advance.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>Correct the value of the constant. If the value for an integer constant is too large, it can also be specified by appending a decimal point as a real constant.</p> <p>Example:                  R1 = 9 876 543 210      correct to:      R1 = 9 876 543 210.</p>
<b>Program continuation by</b>	<p>Press NC START to clear the alarm and continue program execution.</p>
<b>12 180</b>	Channel %1 block %2 illegal chaining of operators %3
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Chained operators</p> <p>The term 'operator chaining' denotes binary and unary operators chained one after another, without the use of brackets.</p> <p>Example:                  N10 R1=R2-(-R3)      ; Correct notation                  N10 R1=R2--R3      ; Error!</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>Formulate the expression correctly and unambiguously using brackets; this increases clarity and eligibility of a program.</p>
<b>Program continuation by</b>	<p>Press NC START to clear the alarm and continue program execution.</p>
<b>12 290</b>	Channel %1 block %2 arithmetic variable %3 not defined
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Source string of arithmetic variable</p> <p>Only the R parameters as arithmetic variables are predefined - the remaining arithmetic variable must be defined using the DEF instruction before they can be used. 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).</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Corrected block.</p>
<b>Remedy</b>	<p>Press NC Stop and select the Corrected Block function using the correct PROGRAM softkey. The correction cursor is positioned on the block to be corrected.</p>

	Set the desired variable in the definition part of the program (if desired, in the calling program if a global variable is to be used).
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 420</b>	Channel %1 block %2 identifier %3 too long
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The symbol to be defined and/or the specified jump target has a name which exceeds the permitted 32 characters.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	The symbol to be created or the jump target for program jumps (label) must be selected within the system conventions, i.e. the name must begin with 2 letters (but the 1st character may not be a \$ character) and may include max. 32 characters.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 450</b>	Channel %1 block %2 label defined repeatedly
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The label of this block exists already. If you compile the NC program offline, 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. (In this case, only the current program execution is compiled, i.e. program branches that are not currently passed will also not be checked and can therefore contain programming errors).
<b>Response</b>	Alarm display. Interface signals are set. Corrected block.
<b>Remedy</b>	Press NC Stop and select the Corrected Block function using the CORRECT PROGRAM. 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 to find out where the searched designation occurs for the first time, and alter one of the two names.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 470</b>	Channel %1 block %2 unknown G function %3 used
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Source string A non-defined G function has been programmed in the displayed block.

<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Use the Programming Instructions of the machine manufacturer to decide whether the displayed G function is generally not available or not possible. Remove the G function from the part program or program the function call with respect of the Programming Instructions of the machine manufacturer.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.

<b>12 490</b>	Channel %1 block %2 access permission level %3 is not valid
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Remove the respective string from the program.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.

<b>12 540</b>	Channel %1 block %2 is too long or too complex
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The max. block length may not exceed 128 characters.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Split the program block into several partial blocks.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.

<b>12 550</b>	Channel %1 block %2 identifier %3 not defined or option does not exist
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Source string The displayed identifier is not defined in the control system.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Press NC STOP and check the machine data file in the NC program or check the faulty block. <ul style="list-style-type: none"> <li>• Correct the name used (type error)</li> <li>• Check the options.</li> </ul>



<b>Program continuation by</b>	Press NC START to restart the program or re-import the machine data file.
<b>12 560</b>	Channel %1 block %2 programmed value %3 exceeds allowed limits
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Source string The programmed numerical value is out of the allowed limits.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Adhere to the value range.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>12 600</b>	Channel %1 block %2 invalid Checksum of Line
<b>Explanation</b>	%1 = Channel number %2 = Block number When executing an INI or TEA file, an invalid line check sum has been detected.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Correct the INI file or correct the MD and create a new INI file (via 'upload').
<b>Program continuation by</b>	Power ON
<b>14 000</b>	Channel %1 block %2 error at end of file
<b>Explanation</b>	%1 = Channel number %2 = Block number, label M02 or M30 is expected as the file end of main programs, and M17 as the file end for subroutines. No subsequent block is provided from the block preparation (data management) although no file end has been programmed in the preceding block.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>14 001</b>	Channel %1 block %2 error at end of block
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>A system-internal data manipulation (e.g. reloading from an external) can be followed by a partial file, without LF as the last character.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>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.</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>14 011</b>	Channel %1 block %2 called program does not exist or is not released
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>The called program (main program or subroutine) has been called from the currently running part program (main program or subroutine). However, either it does not exist in the NC memory, or the option for the used function is not enabled.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Correct the NC part program.</p> <ol style="list-style-type: none"> <li>1. Check the subroutine name in the calling program.</li> <li>2. Check the name of the called program</li> <li>3. Check whether the program has been transferred into the NC memory.</li> <li>4. Check options or upgrade/enable.</li> </ol>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>14 012</b>	Channel %1 block %2 lowest subroutine level exceeded
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>The maximum nesting depth of 4 program levels has been exceeded.</p> <p>Subroutines that have a maximum nesting depth of 3 subroutines can be called from the main program.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p>

	Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Modify 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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 013</b>	Channel %1 block %2 number of subroutine passes invalid
<b>Explanation</b>	%1 = Channel number %2 = Block number, label When a subroutine is called, the programmed number of passes P is either zero or negative.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Program a number of passes between 1 and 9 999.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 014</b>	Channel %1 selected program or access permission not available
<b>Explanation</b>	%1 = Channel number The selected part program is not in NC memory.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Reload the desired program into the NC memory or check the name of the directory (workpiece overview) and correct it.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. Restart part program.
<b>14 015</b>	Channel %1: no access permission for file
<b>Explanation</b>	%1 = Channel number The user has no execution rights for the file.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Alter user rights
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>14 040</b>	Channel %1 block %2 error in end point of circle
<b>Explanation</b>	<p>%1 = Channel number %2 = Block number, label</p> <p>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.</p> <ol style="list-style-type: none"> <li>1. Radii: The NCK calculates the radii for start and end point from the current start point and the remaining programmed circle parameters. The alarm message is output if the difference of the circle radii is greater than the value in MD 21 000 CIRCLE_ERROR_CONST.</li> <li>2. Center points: If the circles have a permissible difference, it is checked whether the circle center point on the mean perpendicular is between start and end point.</li> </ol>
<b>Response</b>	<p>Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.</p>
<b>Remedy</b>	Check MD 21 000 CIRCLE_ERROR_CONST. If the values are within reasonable limits, the circle end point or the circle center point of the part program block must be programmed more exactly.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 050</b>	Channel %1 block %2 nesting depth for arithmetic operations exceeded
<b>Explanation</b>	<p>%1 = Channel number %2 = Block number, label</p> <p>To calculate arithmetic expressions in NC block, an operand stack of fixed size is used. With very complex expressions, this stack might overflow.</p>
<b>Response</b>	<p>Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.</p>
<b>Remedy</b>	Distribute complex arithmetic expressions over several, more simply structured arithmetic blocks.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 051</b>	Channel %1 block %2 arithmetic error in part program
<b>Explanation</b>	<p>%1 = Channel number %2 = Block number, label</p> <ul style="list-style-type: none"> <li>• With the calculation of an arithmetic expression, an overflow occurred (e.g. division by zero).</li> <li>• The value range that can be represented for a certain data type has been exceeded.</li> </ul>

<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Analyze the program and correct the faulty program section.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 080</b>	Channel %1 block %2 jump destination not found
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  The jump target of conditioned and unconditioned jumps within 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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Check the NC program for the following possible errors: <ol style="list-style-type: none"> <li>1. Check whether the target designation is identical to the label.</li> <li>2. Is the jump direction correct?</li> <li>3. Has the label been completed with a colon?</li> </ol>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 092</b>	Channel %1 block %2 axis %3 has wrong axis type
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Correct part program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 095</b>	Channel %1 block %2 circle programmed with zero radius
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  Under the keyword CR=.. for the circle radius, value "0" has been programmed.

<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Circle radius less than CR=.. positive (circle is less than or equal to semi-circle) CR=.. negative (circle is greater than a semi-circle)
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 130</b>	Channel %1 block %2 too many initialization values given
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Reduce the number of initialization values.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 600</b>	Channel %1 block %2 buffer for sequential reload cannot be established
<b>Explanation</b>	%1 = Channel number %2 = Block number, label When loading the INITIAL_INI block, the reload buffer could not be established because of insufficient memory in the RAM of the NCK.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Create free space in the NCK area, for example, by deleting part programs no longer used.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 601</b>	Channel %1 block %2 reload buffer cannot be cleared
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The reload buffer for "Execution from external" could not be cleared; probable cause: <ul style="list-style-type: none"> <li>• MMC-PLC communication not completed.</li> </ul>

<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	All reload buffers are cleared on Power On.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 710</b>	Channel %1 block %2 error during phase %3 of INIT block generation
<b>Explanation</b>	%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 in synchronization (program advance/main run). Section 1:       Error when selecting tool length compensation. Section 2:       Error when selecting transformation. Section 3:       Error when selecting zero offset.  During booting, the cycle interfaces are additionally read in. If an error occurs during this process, "Section 5" is signaled.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	In section 0-3:   Load standard machine data In section 5:    Reload cycles
<b>Program continuation by</b>	Restart
<b>14 750</b>	Channel %1 block %2 too many auxiliary functions programmed
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  More than 10 auxiliary functions have been programmed in a block.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>14 760</b>	Channel %1 block %2 auxiliary function of a group programmed repeatedly
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>The M functions can be divided into groups via machine data completely variable if required. Auxiliary functions are grouped such that several individual functions of a group mutually exclude. Within a group, only one auxiliary function is reasonable and permitted.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Program only one auxiliary function or auxiliary function group. (For group division see Programming Instructions of the machine manufacturer.)
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 770</b>	Channel %1 block %2 auxiliary function programmed incorrectly
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>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).</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Correct the part program - max. 16 auxiliary functions, max. 5 M functions per NC block, max. 1 auxiliary function per group.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 780</b>	Channel %1 block %2 unreleased option used
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>An option not released is used in the block</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Modify part program, upgrade option.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.



<b>14 800</b>	Channel %1 block %2 programmed path speed less or equal to zero
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>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.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	Program the path velocity (geometrical sum of the velocity components of the geometry axes involved) within the limits specified above.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 820</b>	Channel %1 block %2 negative value for maximum spindle speed programmed with constant cutting speed
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>You can program a maximum spindle speed for the function "Constant cutting speed G96" using the key word LIMS=... The value range is between 0.1 - 999 999.9 [rev/min].</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	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 speed or in the block.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 830</b>	Channel %1 block %2 wrong feed type selected
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>G97 has been programmed in the displayed block although G96 (or already G97) has not been activated.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>

**Remedy** Remove G97 from the displayed block and program the correct feed type (G94, G95 or G96) for the next following machining section.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**14 840** Channel %1 block %2 value for constant cutting speed out of range

**Explanation** %1 = Channel number  
 %2 = Block number, label  
 The programmed cutting speed 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]

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

**Remedy** Program the cutting speed under address S within the permissible value range.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**14 900** Channel %1 block %2 use either center point or end point programming

**Explanation** %1 = Channel number  
 %2 = Block number, label  
 When the circle was programmed using the opening angle, the circle center point and, in addition, the circle end point has been programmed, too. The circle is thus 'overdetermined'. Only one of the two points is permitted.

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

**Remedy** Select that programming variant at which the dimensions can be taken over from the workpiece drawing without doubts (in order to avoid errors in calculation).

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**14 910** Channel %1 block %2 invalid angle of aperture for programmed circle

**Explanation** %1 = Channel number  
 %2 = Block number, label  
 When programming the circle using the opening angle, a negative opening angle or an opening angle  $\geq 360$  degrees has been programmed.

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

<b>Remedy</b>	Program the opening angle within the permitted value range of 0.0001 - 359.9999 [degrees].
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>14 920</b>	Channel %1 block %2 intermediate point of circle incorrect
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>15 170</b>	Channel %1 block %2 program %3 could not be compiled
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = String  An error occurred in compilation mode. The (compiler) message displayed after the error refers to the program specified here.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Correct the part program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>15 175</b>	Channel %1 block %2 program %3 interfaces could not be built
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = String  An error occurred during the interface creation mode. The (compiler) error message displayed after the error occurred refers to the program specified here.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Correct the part program.

<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>15 180</b>	Channel %1 block %2 program %3 cannot be executed as INI file
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = String Errors occurred when reading in data as an INI file. The displayed error message refers to the program specified here.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Correct the part program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>15 185</b>	Channel %1 %2 Errors in INI file
<b>Explanation</b>	%1 = Channel number %2 = Number of detected errors Errors have been detected when processing an INI file.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Correct INI file or MD and create new INI file (via 'upload').
<b>Program continuation by</b>	Power ON
<b>15 300</b>	Channel %1 block %2 invalid number-of-passed-blocks during block search
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Enter only positive numbers of passes within the value range.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>15 320</b>	Channel %1 block %2 invalid block search command
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The search job (type of searching target) is less than "1" or greater than "5". It is entered in the Type column of the block 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
<b>Response</b>		Alarm display.
<b>Remedy</b>		Alter search job.
<b>Program continuation by</b>		Press the Clear key to clear the alarm. No further operation required.
<b>15 330</b>		Channel %1 block %2 invalid block number as target of block search
<b>Explanation</b>		%1 = Channel number %2 = Block number, label  Syntax error! Only positive integer numbers are permitted as block numbers. For main blocks, ":" must be written in front of them, and an "N" must be written in front of auxiliary blocks.
<b>Response</b>		Alarm display.
<b>Remedy</b>		Repeat your entry with the corrected block number.
<b>Program continuation by</b>		Press the Clear key to clear the alarm. No further operation required.
<b>15 340</b>		Channel %1 block %2 invalid label as target of block search
<b>Explanation</b>		%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.
<b>Response</b>		Alarm display.
<b>Remedy</b>		Repeat your entry with the corrected label.
<b>Program continuation by</b>		Press the Clear key to clear the alarm. No further operation required.
<b>15 350</b>		Channel %1 block %2 target of block search not found
<b>Explanation</b>		%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.
<b>Response</b>		Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>		Check the part program, alter the search target (type error in part program) and restart search.

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>15 370</b>	Channel %1 target of block search not found
<b>Explanation</b>	%1 = Channel number An invalid search target (e.g. negative block number) has been specified for block search.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check specified block number, label or string. Repeat your entry with the correct search target.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>15 400</b>	Channel %1 block %2 selected initial ini file does not exist
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The operator has selected an INI block for a read, write or processing function, which: <ol style="list-style-type: none"> <li>1. does not exist in the NCK area, or</li> <li>2. which does not have the required protection level required to execute the function.</li> </ol>
<b>Response</b>	Alarm display.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>15 410</b>	Channel %1 block %2 initialization file contains invalid M function
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>15 420</b>	Channel %1 block %2 instruction not accepted in current mode
<b>Explanation</b>	%1 = Channel number %2 = Block number, label When executing an Init block, the interpreter has found an illegal instruction (e.g. traversing instruction).
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>15 460</b>	Channel %1 block %2 syntax conflict with modal G functions
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The addresses programmed in the block are not compatible with the modal, syntax-defining G function. Example: N100 G01 .. I . J. K. LF
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Correct the displayed block; match G functions and addresses in the block with each other.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>15 800</b>	Channel %1 block %2 wrong starting condition for CONTPRON
<b>Explanation</b>	%1 = Channel number %2 = Block number, label The start conditions for contour preparation (LCYC 95) are faulty: G40 (deselection of tool radius compensation) is not active
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Modify part program: Deselect tool radius compensation with G40.

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 110</b>	Channel %1 block %2 spindle %3 for dwell time not in speed control mode
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Axis, spindle</p> <p>The spindle can be in the following modes: positioning mode, oscillation mode, control mode.</p> <p>Positioning mode:                  Position control (spindle position under SPOS)</p> <p>Oscillation mode:                  Speed control (M41 - M45 or M40 and S..)</p> <p>Control mode:                  Speed control (spindle speed under S.., M3/M4/M5)</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	<p>Check part program for correct spindle number.                  Switch the desired spindle to control mode before calling the dwell time with M3, M4 or M5.</p>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 420</b>	Channel %1 block %2 axis %3 repeatedly programmed
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Axis name, spindle number</p> <p>It is not allowed to program an axis several times.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Delete addresses programmed several times.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 500</b>	Channel %1 block %2 chamfer or radius negative
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label</p> <p>A negative chamfer or rounding has been programmed under the key words CHF= .., RND=...</p>



<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Use only positive values to program chamfers, roundings and modal roundings.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 510</b>	Channel %1 block %2 facing axis is not defined
<b>Explanation</b>	%1 = Channel number %2 = Block number, label Diameter programming has been programmed without facing axis. This function is not available for turning/milling.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Remove diameter programming from the NC program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 700</b>	Channel %1 block %2 axis %3 invalid feed type
<b>Explanation</b>	%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. <ol style="list-style-type: none"> <li>1. G33 (thread with constant lead) and feed has not been programmed with G94 or G95.</li> <li>2. G33 (thread with constant lead) is active (maintained) and, in addition, G63 is programmed in a following block → conflict! (G63 is in the 2nd, G33 in the 1st group).</li> </ol>
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Use only feed type G94 or G95 for thread cutting functions. Use G01 to deselect the thread cutting function after G33 and in front of G63.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>16 710</b>	Channel %1 block %2 axis %3 master spindle not programmed
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Axis name, spindle number</p> <p>A master spindle function has been programmed (G33, G95, G96), without programming the speed or the direction of rotation of the master spindle.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Add S value or direction of rotation for the master spindle in the displayed block.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 715</b>	Channel %1 block %2 axis %3 master spindle not in standstill
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Spindle number</p> <p>The spindle must not move when the function is in use (G74, reference point approach).</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	Program M5 or SPOS in the part program in front of the faulty block.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 720</b>	Channel %1 block %2 axis %3 thread lead is zero
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Block number, label                  %3 = Axis name, spindle number</p> <p>No lead has been programmed in a thread block with G33 (thread with constant lead).</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  Interpreter stop.                  NC Start inhibited.</p>
<b>Remedy</b>	<p>The thread lead for the specified geometry axis must be programmed under the associated interpolation parameter.</p> <p>X I                  Y J                  Z K</p>

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 730</b>	Channel %1 block %2 axis %3 wrong parameter for thread cutting
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number During thread cutting with G33, the parameter has not been specified for the velocity-defining axis (axis with the longest traversing path).
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Program the thread lead under the appropriate Ipo (= interpolation) program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 740</b>	Channel %1 block %2 geometry axis must be programmed
<b>Explanation</b>	%1 = Channel number %2 = Block number, label No geometry axis has been programmed for thread cutting (G33). However, the geometry axis is absolutely necessary when an interpolation parameter was specified.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Specify geometry axis and respective interpolation parameter.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 760</b>	Channel %1 block %2 axis %3 S value missing
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number The spindle speed for tapping without compensation chuck (G331 or G332) has not been specified.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Program spindle speed under address S in [rev/min] (despite of axis mode); the direction of rotation results from the spindle lead.

	positive thread lead:	Direction of rotation as with M03
	negative thread lead:	Direction of rotation as with M04
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.	
<b>16 762</b>	Channel %1 block %2 spindle %3 function of thread or drill is active	
<b>Explanation</b>	%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.	
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.	
<b>Remedy</b>	Modify part program. Deselect thread cutting or tapping.	
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.	
<b>16 763</b>	Channel %1 block %2 axis %3 programmed speed is illegal (zero or negative)	
<b>Explanation</b>	%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.	
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.	
<b>Remedy</b>	The programmed speed (S value) must be positive. Depending on the particular application, zero can also be accepted (e.g. G25 S0).	
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.	
<b>16 770</b>	Channel %1 block %2 axis %3 encoder missing	
<b>Explanation</b>	%1 = Channel number	
	%2 = Block number, label	
	%3 = Axis name, spindle number	
	A position was programmed for the axis which requires a measuring system. According to MD 30 200 NUM_ENC_S, this machine axis has no measuring system.	
<b>Response</b>	Alarm display. Interface signals are set.	

	Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Remove the respective function (e.g. SPOS) from the part program, or enter an existing measuring system in MD 30 200 NUM_ENC_S.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 830</b>	Channel %1 block %2 invalid position for axis/spindle %3 programmed
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Program a position within the range 0 - 359.999.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>16 903</b>	Channel %1 program control: action %2 not allowed in the current state
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) The action concerned cannot be processed at the moment. This can occur when importing machine data, for example.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Wait until the previous operation is completed, or cancel the operation with reset and repeat the action.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 904</b>	Channel %1 program control: action %2 not allowed in the current state
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) Processing (program, Jog, block search, reference point, ...) cannot be started or continued in the current state.
<b>Response</b>	Alarm display
<b>Remedy</b>	Check program state and channel state.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>16 905</b>	Channel %1 program control: action %2 not allowed
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Action number/action name (see Section 1.4 Action List)</p> <p>Processing cannot be started or continued. A start will only be accepted if an NCK function can be started.</p> <p>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.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check program state and channel state.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 906</b>	Channel %1 program control: action %2 is aborted because of an active alarm
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Action number/action name (see Section 1.4 Action List)</p> <p>The action has been canceled due to an alarm.</p> <p>Processing cannot be started or continued. A start will only be accepted if an NCK function can be started.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Eliminate the error and acknowledge the alarm. Then restart the process.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 907</b>	Channel %1 action %2 only possible in stop
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Action number/action name (see Section 1.4 Action List)</p> <p>The action may only be carried out in Stop condition.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check program state and channel state.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 908</b>	Channel %1 action %2 only possible in reset or at the block end
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Action number/action name (see Section 1.4 Action List)</p> <p>Action %2 may only be carried out in reset state or at block end.</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check program state and channel state.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>16 909</b>	Channel %1 the action %2 is not allowed in current mode
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) Another mode must be activated for the activated function.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check operation and operating mode.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 911</b>	Channel %1 mode change is not allowed
<b>Explanation</b>	%1 = Channel number Changing from overstore to another operating mode is not allowed. The 2nd parameter specified describes the recommended operating mode to be selected.
<b>Response</b>	Alarm display.
<b>Remedy</b>	After you have quitted the Overstore mode, you can change to another mode.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 912</b>	Channel %1 program control: action %2 only possible in reset
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) This action can only be carried out in Reset condition.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Press Reset or wait until processing is completed.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 913</b>	BAG %1 channel %2 mode change: action %3 not allowed
<b>Explanation</b>	%1 = Channel number %2 = Mode group number %3 = Action number/action name (see Section 1.4 Action List) Changing to the desired mode is not allowed. The change 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!
<b>Response</b>	Alarm display.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

<b>16 914</b>	Mode group %1 channel %2 mode change: action %3 not allowed
<b>Explanation</b>	%1 = Channel number %2 = Mode group number %3 = Action number/action name (see Section 1.4 Action List) Wrong mode change, e.g.: AUTO → MDAREF
<b>Response</b>	Alarm display.
<b>Remedy</b>	Check operation or selected mode.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 915</b>	Channel %1 action %2 in the current block not allowed
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) Reorganization of block processing is currently not possible. Mode change cannot be carried out. The 2nd parameter describes the action recommended to interrupt block execution.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Continue program execution up to a reorganized NC block or modify part program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 916</b>	Channel %1 reposition: action %2 not allowed in the current state
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) Repositioning of block execution is currently not possible. Mode change can therefore not be carried out. The 2nd parameter described the action recommended to use for repositioning.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Continue the program to an NC block which can be repositioned, or modify part program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 918</b>	Channel %1: for action %2 needs reset in all channel.
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) To execute the action, the channel must be in initial state! (e.g. for loading _N_INITIAL_INI)
<b>Response</b>	Alarm display.
<b>Remedy</b>	Press the RESET key.



<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 919</b>	Channel %1 action %2 is not allowed, because of an alarm
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) Action %2 cannot be executed due to an alarm or the channel is in error condition.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Press the RESET key.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 920</b>	Channel %1 action %2 is already in enabled
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 Action List) An identical action is still being executed.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Wait until the previous operation is completed and then repeat the action..
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>16 922</b>	Channel %1 subprograms: action %2 maximum stack level exceeded
<b>Explanation</b>	%1 = Channel number %2 = Action number/action name (see Section 1.4 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.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited. NC Stop at alarm.
<b>Remedy</b>	Press the RESET key. Before starting the program, check the program nesting depth and reduce it or avoid interruptions. Example: It is not recommended to interrupt the approach block of a repositioning process.

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**16 923** Channel %1 program control: action %2 not allowed in the current state

**Explanation** %1 = Channel number

%2 = Action number/action name (see Section 1.4 Action List)

Current program execution cannot be stopped, as merely one block search operation is active.

For example, this applies to the loading of machine data and to block search until the searched target has been found.

**Response** Alarm display.

Interface signals are set.

**Remedy** Press Reset to cancel!

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**16 924** Channel %1 caution: program test will change the tool 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 back-up copy of his data or to re-copy the data on completion of this process.

**Response** Alarm display.

**Remedy** Save the tool data to MMC and recopy them when the program test is completed.

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**16 925** Channel %1 program control: action %2 not allowed in the current state

**Explanation** %1 = Channel number

%2 = Action number/action name (see Section 1.4 Action List)

The action has been denied, since a process change is being carried out. Process change means mode change, switching on or off Overstore.

Example:  
NC Start during mode change from AUTOMATIC to MDA.

**Response** Alarm display.

**Remedy** Repeat the command.

**Program continuation by** Press the Clear key to clear the alarm. No further operation required.

**16 931** Channel %1 subprogram: action %2 maximum stack level exceeded

**Explanation** %1 = Channel number

%2 = Action number/action name (see Section 1.4 Action List)

The current operation can be interrupted by various actions.

	<p>Depending on the action, control-internal programs are activated. These programs can be interrupted in similar fashion as the NC program. For memory reasons, any nesting depth of the control-internal programs is not possible.</p> <p>Example: Do not interrupt the approach block of a repositioning process repeatedly, but wait until the approach block is executed.</p> <p>Possible actions are mode change, block suppression, overstore, etc.</p>
<b>Response</b>	Alarm display
<b>Remedy</b>	Initiate block change and repeat the action.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>17 001</b>	Channel %1 block %2 no memory left for tool or magazine data
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>The number of the tool data in the NC is limited.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	Delete tools not used.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 010</b>	Channel %1 block %2 no memory left for symbol
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>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).</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Interpreter stop.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	Make more memory space available for subroutine calls and tool offsets.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 020</b>	Channel %1 block %2 1st array index out of range
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Block number, label</p> <p>A read or write access to a field variable (e.g. arithmetic parameter) with invalid 1st field index has been programmed.</p> <p>For example, R2000 = 5; parameter number 2000 not defined.</p>

**Response** 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** Press the RESET key to clear the alarm. Restart part program.

**17 070** 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.

**Response** 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** Press the RESET key to clear the alarm. Restart part program.

**17 080** Channel %1 block %2 value violates 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.

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

**Remedy** Determine the input limits of the MD and carry out value assignment within these limits.

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**17 090** Channel %1 block %2 value violates 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.

**Response** Alarm display.  
Interface signals are set.

	Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Determine the input limits of the MD and carry out value assignment within these limits.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 160</b>	Channel %1 block %2 tool is not selected
<b>Explanation</b>	%1 = Channel number %2 = Block number, label You have tried to access the current tool compensation data although no tool has been selected.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Program or activate a tool compensation in the NC part program. Example: N100 G. .. T5 D1 .. LF
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 180</b>	Channel %1 block %2 illegal D number
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Check tool call in NC part program: <ul style="list-style-type: none"> <li>• Are the correct edge number (D number) programmed? If no edge number is specified, D1 is automatically active.</li> <li>• All tool parameters defined? The dimensions of the tool edge must have been entered in advance either via the operator panel or the V24 interface.</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 190</b>	Channel %1 block %2 illegal T number
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Check tool call in NC part program: <ul style="list-style-type: none"> <li>• Is the correct tool number T... programmed?</li> <li>• Tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered in advance either via the operator panel or the V24 interface.</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 200</b>	Channel %1 block %2: cannot delete an active tool
<b>Explanation</b>	%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. This applies both to the tool preselected and changed by T, and for tools for which a constant wheel circumferential speed or tool monitoring is active.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Deselect tool
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>17 220</b>	Channel %1 block %2 tool not available
<b>Explanation</b>	%1 = Channel number %2 = Block number, label  Whenever you try to access a tool that has not (yet) been defined using a T number.
<b>Response</b>	Alarm display. Interface signals are set. Interpreter stop. NC Start inhibited.
<b>Remedy</b>	Correct the NC program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>20 000</b>	Channel %1 axis %2 reference cam not reached
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Axis name, spindle number</p> <p>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 in the case of incremental encoders).</p>
<b>Response</b>	<p>NC Stop at alarm.</p> <p>NC Start inhibited.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>This may have three possible error causes:</p> <ol style="list-style-type: none"> <li>1. The value in MD 34 030 REFP_MAX_CAM_DIST is too low. 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 MD.</li> <li>2. The cam signal does not pass to 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).</li> <li>3. The reference-point switch is not actuated by the cam. Check the vertical distance between reducing cam and actuating switch.</li> </ol>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>20 001</b>	Channel %1 axis %2 cam signal missing
<b>Explanation</b>	<p>%1 = Channel number</p> <p>%2 = Axis name, spindle number</p> <p>At the beginning of phase 2 of reference-point approach, the signal from the reducing cam is no longer available.</p> <p>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.</p>
<b>Response</b>	<p>NC Stop at alarm.</p> <p>NC Start inhibited.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>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.</p> <p>When the axis has stopped on the cam, check whether the signal "DELAYED REFERENCE-POINT APPROACH" (V380x1000.7) is still provided.</p> <ul style="list-style-type: none"> <li>• Hardware: wire break? short circuit?</li> <li>• Software: user program?</li> </ul>

**Program continuation by** Press the RESET key to clear the alarm. Restart part program.

**20 002**

**Explanation**

Channel %1 axis %2 zero reference mark not found

%1 = Channel number

%2 = Axis name, spindle number

The zero mark of the incremental position encoder is not within the defined path.

The 2nd phase of reference-point approach ends when the zero mark of the encoder has been detected after the rising/falling edge of the PLC interface signal "DELAYED REFERENCE-POINT APPROACH" (V 380x1000.7) has initiated trigger start. The maximum path distance between trigger start and the zero mark following after the trigger start 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! (Faulty cam adjustment or too high delay by PLC user program).

**Response**

NC Stop at alarm.

NC Start inhibited.

Alarm display.

Interface signals are set.

**Remedy**

Check 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**

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

**20 004**

**Explanation**

Channel %1 axis %2 reference mark missing

%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 REFP\_MAX\_MARKER\_DIST expected within the 2 reference marks counts from the start point.

**Response**

NC Stop at alarm.

NC Start inhibited.

Alarm display.

Interface signals are set.



<b>Remedy</b>	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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>20 005</b>	Channel %1 axis %2 reference point approach aborted
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number The channel-specific referencing could not be completed for all axes specified (e.g. cancellation due to missing servo enable, measuring system change, releasing the direction key, etc.).
<b>Response</b>	NC Stop at alarm. NC Start inhibited. Alarm display. Interface signals are set.
<b>Remedy</b>	Check cancel facilities: <ul style="list-style-type: none"> <li>• Servo enable missing (V 380x0001.1)</li> <li>• Direction keys + or - missing (V 380x0004.6 and .7)</li> <li>• Feed override = 0</li> </ul> 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.).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>20 006</b>	Channel %1 axis %2 reference point creep velocity not reached
<b>Explanation</b>	%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). Phase 2 is canceled (now in front of the cam), and reference-point approach is automatically restarted with phase 1. If the approach velocity is not yet reached even with the 2nd try, referencing is finally aborted, and a respective alarm message is output. Approach speed: MD 34 040 REFP_VELO_SEARCH_MARKER Speed tolerance: MD 35 150 SPIND_DES_VELO_TOL

<b>Response</b>	NC Stop at alarm. NC Start inhibited. Alarm display. Interface signals are set.
<b>Remedy</b>	Reduce 34 040 REFP_VELO_SEARCH_MARKER and/or increase the MD for the speed tolerance MD 35 150 SPIND_DES_VELO_TOL.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>20 050</b>	Channel %1 axis %2 handwheel mode active
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	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).
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.
<b>20 051</b>	Channel %1 axis %2 handwheel mode not possible
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Decide whether you want to traverse the axis via the direction keys or via the handwheel.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.
<b>20 055</b>	Channel %1 Master spindle does not exist in JOG mode
<b>Explanation</b>	%1 = Channel number You wish to traverse at revolutional feedrate in the JOG mode, but there is no master spindle.
<b>Response</b>	Alarm display. Interface signals are set.
<b>Remedy</b>	Define the master spindle.
<b>Program continuation by</b>	Press the cancel key to clear the alarm. No further operation required.

<b>20 056</b>	JOG: Channel %1 axis %2 no revolutional feedrate possible. Axis/spindle %3 stationary
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number %3 = Axis name, spindle number An axis is to be traversed in JOG mode with feedrate per revolution. However, the feed of the spindle/axis from which the feed is to be derived is zero.
<b>Response</b>	Alarm display.
<b>Remedy</b>	The spindle/axis from which the feed is to be derived is to be traversed.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause. No further operation required.
<b>20 057</b>	Channel %1 block %3 Revolutional feedrate for axis/spindle %2 is <= 0
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label Revolutional feedrate has been programmed for an axis/spindle, but no velocity/speed was specified, or the programmed value is less than zero.
<b>Response</b>	Alarm display. Interface signals are set. NC Stop at alarm. NC Start inhibited. Channel processing not ready.
<b>Remedy</b>	Correct the part program accordingly.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>20 062</b>	Channel %1 axis %2 already active
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number The axis to be traversed cannot be traversed, since it is already being traversed.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Start the geometry axis only when the traversing movement as a machine axis is completed.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>21 612</b>	Channel %1 axis %2 VDI signal 'drive enable' reset during traverse motion
<b>Explanation</b>	%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.

<b>Response</b>	NC Start inhibited. Alarm display. Interface signals are set. NC Stop at alarm.
<b>Remedy</b>	Check interface signal "Servo Enable" (V 380x0002.1). Trace the signal back to the sections in the PLC user program from which the signal is linked and set/ deleted.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>21 614</b>	Channel %1 axis %2 hardware limit switch %3 reached
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. NC Start inhibited.
<b>Remedy</b>	<ol style="list-style-type: none"> <li>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).</li> <li>2. If the axis has not yet been approached to the reference point, you can leave the hardware limit switch in JOG mode in the opposite direction.</li> <li>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.</li> </ol>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>22 000</b>	Channel %1 block %3 spindle %2 change of gear stage not possible
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display. Interface signals are set. NC Stop at alarm. NC Start inhibited.

<b>Remedy</b>	Change to control mode of spindle in front of the S word that requires gear stage change. The change to the control mode is carried out with: <ul style="list-style-type: none"> <li>• M03, M04, M05 or M41 .. M45 from axis mode and positioning mode</li> <li>• interface signal "Gear changed" (V 38032000.3) from oscillation drive</li> </ul>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 010</b>	Channel %1 block %3 spindle %2 actual gear stage differs from requested gear stage.
<b>Explanation</b>	%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.
<b>Response</b>	Alarm display.
<b>Remedy</b>	Correct the PLC program.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>22 050</b>	Channel %1 block %3 spindle %2 transition from speed control mode to position control mode not possible
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label Transition from speed control to position control is not possible: When position control is switched on, the spindle speed is greater than the limit speed of the measuring system.
<b>Response</b>	NC Start inhibited. NC Stop at alarm. Alarm display. Interface signals are set.
<b>Remedy</b>	Preset lower speed (S value) prior to switching on. Sacrifice of SPOS, SPOSA and SPCON, or connect measuring system.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 053</b>	Channel %1 block %3 spindle %2 reference mode not supported
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label

<b>Response</b>	NC Start inhibited. NC Stop at alarm. Alarm display. Interface signals are set.
<b>Remedy</b>	Change setting of MD 34 200 ENC_REFP_MODE, change to JOG + REF and then approach reference point. Correct PLC program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 062</b>	Channel %1 axis %2 reference point approach: search speed for zero mark (MD) is not reached
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number The configured zero mark search speed is not reached.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited NC Stop at alarm
<b>Remedy</b>	Check active speed limiting. Configure lower zero mark search speed in MD 34 040 REFP_VELO_SEARCH_MARKER. Check the tolerance range for the actual speed MD 35 150 SPIND_DES_VELO_TOL. Set another referencing mode MD 34 200 ENC_REFP_MODE.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 064</b>	Channel %1 axis %2 reference point approach: search speed for zero mark (MD) is too high
<b>Explanation</b>	%1 = Channel number %2 = Axis name, spindle number The configured zero mark search speed is too high. The encoder frequency for the active measuring system is exceeded.
<b>Response</b>	Alarm display. Interface signals are set. NC Start inhibited NC Stop at alarm
<b>Remedy</b>	Check active speed limitation. Configure lower zero mark search speed MD 34 040 REFP_VELO_SEARCH_MARKER. Check encoder limit frequency configuration MD 36 300 ENC_FREQ_LIMIT and MD 36 302 ENC_FREQ_LIMIT_LOW. Set another referencing mode MD 34 200 ENC_REFP_MODE
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>22 100</b>	Channel %1 block %3 spindle %2 chuck speed exceeded
<b>Explanation</b>	<p>%1 = Channel number          %2 = Axis name, spindle number          %3 = Block number, label</p> <p>The actual speed of the displayed spindle is greater than defined in the axis-specific MD 35 100 SPIND_VELO_LIMIT plus the tolerance set in MD 35 150 SPIND_DES_VELO_TOL.</p> <p>Correct optimization by the drive manufacturer provided, this alarm cannot occur!</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>Mode group ('BAG') not ready for operation.</p> <p>Channel not ready for operation.</p> <p>NC Stop at alarm.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Check start-up and optimization data with respect of the Start-Up Instructions of the drive manufacturer and correct them.</p> <p>Increase tolerance window in MD 35 150 SPIND_DES_VELO_TOL.</p>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 101</b>	Channel %1 block %3 spindle %2 maximum speed for encoder resynchronization exceeded
<b>Explanation</b>	<p>%1 = Channel number          %2 = Axis name, spindle number          %3 = Block number, label</p> <p>The limit frequency of the active encoder (axis-specific MD 36 300 ENC_FREQ_LIMIT) has been exceeded for one of the functions G33 (thread cutting with encoder), G95 (revolution feedrate) or G96 (constant cutting speed). The spindle synchronization got lost as a result of this.</p>
<b>Response</b>	<p>Alarm display.</p> <p>Interface signals are set.</p> <p>NC Stop at alarm.</p> <p>NC Start inhibited.</p>
<b>Remedy</b>	<p>Check the limit frequency default data specified for this encoder in MD 36 300 ENC_FREQ_LIMIT.</p> <p>Check the maximum spindle speed in the axis-specific MD 35 130 GREAR_STEP_MAX_VELO_LIMIT and correct (reduce) it if necessary.</p> <p>Program an upper spindle speed limitation with G26 S.. in the preceding NC block which must be below the maximum encoder limit frequency.</p>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>22 200</b>	Channel %1 block %3 spindle %2 axis stop during tapping
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name, spindle number                  %3 = Block number, label</p> <p>The drilling axis has been stopped via the NC/PLC interface during tapping with compensation chuck (G63) - the spindle continues rotating. The thread and (in some cases) also the tapping drill have been damaged as a result of this.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  NC Stop at alarm.                  NC Start inhibited.</p>
<b>Remedy</b>	Provide for 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.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 250</b>	Channel %1 spindle %2 axis stopped during thread cutting
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name, spindle number</p> <p>The thread cutting axis has been stopped during an active thread block. The stop can be caused by VDI signals resulting in feed interruption.</p>
<b>Response</b>	<p>Alarm display.                  Interface signals are set.                  NC Start inhibited.</p>
<b>Remedy</b>	Check the axis/spindle-specific stop signals (V 380x0004.3).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>22 260</b>	Channel %1 spindle %2 thread might be damaged
<b>Explanation</b>	<p>%1 = Channel number                  %2 = Axis name                  %3 = Block number</p> <p>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.</p> <p>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!</p>
<b>Response</b>	Alarm display.
<b>Remedy</b>	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.
<b>Program continuation by</b>	Press NC START to clear the alarm and continue program execution.
<b>22 270</b>	Channel %1 block %2 spindle %3 spindle speed too high for thread cutting
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number The spindle speed for thread cutting G 33 is so high that the maximum axis speed is exceeded due to the programmed axis speed..
<b>Response</b>	Alarm display.
<b>Remedy</b>	Program a lower spindle speed or speed limitation with G26 S .. or reduce the spindle speed prior to the thread block via setting data 43 220 SPIND_ MAX_ VELO_ G26 or the spindle override.
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.
<b>25 000</b>	Axis %1 hardware fault of active encoder
<b>Explanation</b>	%1 = Axis name, spindle number The signals of the currently active actual-position encoder are either missing (interface signal V 380x001.5), are not in phase or have ground connection/ short circuit.
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation. NC Start inhibited. NC Stop at alarm. The NC changes to follow-up mode. Alarm display. Interface signals are set. The axes are no longer synchronized with the machine actual value (reference point).
<b>Remedy</b>	Check if the measuring circuit connector is connected correctly. Check encoder signals and replace encoder in case of errors.
<b>Program continuation by</b>	Power ON
<b>25 010</b>	Axis %1 pollution of active encoder
<b>Explanation</b>	%1 = Axis name, spindle number The encoder used for position control signals pollution signal (only with measuring systems that provide this option).
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation.

	<p>NC Start inhibited.</p> <p>NC Stop at alarm.</p> <p>The NC changes to follow-up mode.</p> <p>Alarm display.</p> <p>Interface signals are set.</p> <p>The axes are no longer synchronized with the machine actual value (reference point).</p>
<b>Remedy</b>	Check the measuring system with respect of the settings of the measuring system manufacturer.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>25 020</b>	Axis %1 zero mark monitoring of active encoder
<b>Explanation</b>	<p>%1 = Axis name, spindle number</p> <p>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.</p>
<b>Response</b>	<p>Mode group ('BAG') not ready for operation.</p> <p>Channel not ready for operation.</p> <p>NC Start inhibited.</p> <p>NC Stop at alarm.</p> <p>The NC changes to follow-up mode.</p> <p>Alarm display.</p> <p>Interface signals are set.</p> <p>The axes are no longer synchronized with the machine actual value (reference point).</p>
<b>Remedy</b>	<p>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:</p> <ol style="list-style-type: none"> <li>1. Transmission path: Check the actual value connector on the motor for correct contacting; check encoder cable for continuity, short circuit and ground connection (loose contact?).</li> <li>2. Encoder pulses: Encoder supply within tolerance limits?</li> <li>3. Evaluation electronics: Change/reconfigure the drive module used.</li> </ol> <p>The monitoring can be switched off by setting MD 36 310 ENC_ZERO_MONITORING to "0".</p>
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>25 030</b>	Axis %1 actual velocity alarm
<b>Explanation</b>	<p>%1 = Axis name, spindle number</p> <p>The actual speed 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 36 200 AX_VELO_LIMIT (threshold value for speed monitoring). This threshold value specified in [mm/min, rev/min] is entered by approx. 5 - 10 % greater than it can occur with the maximum traversing speed. Drive errors might result in exceeding the speed what causes the alarm.</p>
<b>Response</b>	<p>Channel not ready for operation.</p> <p>NC Start inhibited.</p> <p>NC Stop at alarm.</p> <p>The NC changes to follow-up mode.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>Check speed setpoint cable (bus cable).</p> <p>Check actual values and position-control direction.</p> <p>Change position-control direction if the axis rotates uncontrolled → axis-specific MD 32 110 ENC_FEEDBACK_POL = &lt; -1, 0, 1 &gt;.</p> <p>Increase monitoring limit value in MD 36 200 AX_VELO_LIMIT.</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>25 040</b>	Axis %1 standstill monitoring
<b>Explanation</b>	<p>%1 = Axis name, spindle number</p> <p>The NC monitors holding of the position in standstill. The monitoring is started after an axis-specific time set in MD 36 040 STANDSTILL_DELAY_TIME after the interpolation has been completed. It is continuously checked whether the axis remains within its tolerance threshold set in MD 36 030 STANDSTILL_POS_TOL.</p> <p>The following cases are possible:</p> <ol style="list-style-type: none"> <li>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.</li> <li>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.</li> </ol>
<b>Response</b>	<p>Channel not ready for operation.</p> <p>NC Start inhibited.</p> <p>NC Stop at alarm.</p> <p>The NC changes to follow-up mode.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>

**Remedy**

- Check MD 36 040 STANDSTILL\_DELAY\_TIME and MD 36 030 STANDSTILL\_POS\_TOL and increase if necessary.
- Estimate machining forces and reduce by feed reduction/speed increase if necessary.
- Increase clamping pressure.
- Increase gain in the open-loop position control circuit by improved optimization (loop-gain factor MD 32 200 POSCTRL\_GAIN).

**Program continuation by**

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

**25 050**

Axis %1 contour monitoring

**Explanation**

%1 = Axis name, spindle number

The NCK calculates the actual value for each interpolation 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 higher than the amount set in MD CONTOUR\_TOL, the program is aborted and the alarm output.

**Response**

Channel not ready for operation.  
 NC Start inhibited.  
 NC Stop at alarm.  
 The NC changes to follow-up mode.  
 Alarm display.  
 Interface signals are set.

**Remedy**

- Check optimization of the position controller (loop-gain factor in MD 32 200 POSCTRL\_GAIN) to make sure that the axis follows the setpoint setting dead-beat. Otherwise, speed controller optimization must be improved or the loop-gain factor ( $K_v$  factor) be reduced.
- Check 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 of this. 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 speed controller optimization
- Check mechanics (smooth running, rotational masses).

**Program continuation by**

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

**25 060**

Axis %1 desired speed limit

**Explanation**

%1 = Axis name, spindle number

The speed setpoint has exceeded its upper limit MD 36 210 CTRL\_OUT\_LIMIT longer than allowed. The specified axis speed exceeds the motor nominal speed set in MD 32 260 RATED\_VELO.

**Response**

Mode group ('BAG') not ready for operation  
 Channel not ready for operation.

	<p>NC Start inhibited.</p> <p>NC Stop at alarm.</p> <p>The NC changes to follow-up mode.</p> <p>Alarm display.</p> <p>Interface signals are set.</p>
<b>Remedy</b>	<p>Correct setting of the drive controller and normal machining conditions provided, this alarm should not occur.</p> <ul style="list-style-type: none"> <li>• Check actual values: local heavy running of carriage, speed dip and torque peak in case of workpiece/tool contact, traversing to a fixed obstacle, etc.</li> <li>• Check direction of position control: Does the axis move uncontrolled?</li> <li>• Check speed setpoint cable.</li> <li>• Reduce axis speed or increase MD 32 260 RATED_VELO.</li> </ul>
<b>Program continuation by</b>	<p>The stepping-switch motor axes must be re-referenced. Press the RESET key to clear the alarm.</p>
<b>25 070</b>	<p>Axis %1 drift limit exceeded</p>
<b>Explanation</b>	<p>%1 = Axis name, spindle number</p> <p>For analog drives only!</p> <p>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 36 710 DRIFT_LIMIT. The drift value itself is not limited.</p> <p>Automatic drift compensation: MD 36 700 DRIFT_ENABLE = 1</p> <p>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.</p> <p>Manual drift compensation: MD 36 700 DRIFT_ENABLE = 0</p> <p>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.</p>
<b>Response</b>	<p>Alarm display.</p>
<b>Remedy</b>	<p>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).</p>
<b>Program continuation by</b>	<p>Press the Clear key to clear the alarm. No further operation required.</p>
<b>25 080</b>	<p>Axis %1 positioning monitoring</p>
<b>Explanation</b>	<p>%1 = Axis name, spindle number</p> <p>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.</p> <p>Exact stop coarse: MD 36 000 STOP_LIMIT_COARSE</p> <p>Exact stop fine: MD 36 010 STOP_LIMIT_FINE</p>

<b>Response</b>	<p>Channel not ready for operation.          NC Start inhibited.          NC Stop at alarm.          The NC changes to follow-up mode.          Alarm display.          Interface signals are set.</p>
<b>Remedy</b>	<p>Check whether the exact stop limit (coarse and fine) corresponds to the dynamic possibilities of the axes; otherwise, increase the exact stop limit, if necessary, in conjunction with the positioning time defined in MD 36 020 POSITIONING_TIME.          Check speed controller/position controller optimization; select gain as high as possible.          Check setting of the loop-gain factor (<math>K_V</math> factor) (MD 32 200 POSCTRL_GAIN); increase if necessary.</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>25 201</b>	<p>Axis %1 drive fault</p>
<b>Explanation</b>	<p>%1 = Axis name, spindle number          For step-switching motor axes only:          The rotation monitoring error occurred. The step-switching motor lost steps due to overload.</p>
<b>Response</b>	<p>Mode group ('BAG') not ready for operation.          Channel not ready for operation.          NC Start inhibited.          NC Stop at alarm.          Alarm display.          Interface signals are set.          Reference point lost.</p>
<b>Remedy</b>	<p>Check application (accelerations, speeds)</p>
<b>Program continuation by</b>	<p>Press the RESET key to clear the alarm. Restart part program.</p>
<b>26 000</b>	<p>Axis %1 clamping monitoring</p>
<b>Explanation</b>	<p>%1 = Axis name, spindle number          The clamped axis has been pressed from its position. The permissible deviation is defined in the axis-specific MD 36 050 CLAMP_POS_TOL.          Axis clamping is activated by the axis-specific interface signal V 380x0002.3: "Clamping running".</p>
<b>Response</b>	<p>Mode group ('BAG') not ready for operation.          Channel not ready for operation.          NC Start inhibited.          NC Stop at alarm.          The NC changes to follow-up mode.</p>

	Alarm display. Interface signals are set.
<b>Remedy</b>	Determine deviation of actual position to set 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).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>26 002</b>	Axis %1 encoder %2 resolution configured for drive and position controller differ
<b>Explanation</b>	%1 = Axis name, spindle number %2 = Encoder number An illegal value has been entered in MD 31 020 ENC_RESOL, e.g.. Ø.
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation. NC Start inhibited. NC Stop at alarm. The NC changes to follow-up mode. Alarm display. Interface signals are set.
<b>Remedy</b>	For step-switching motors: The number of encoder marks specified in MD 31 020 ENC_RECOL must be the same as in MD 31 400 STEP_RESOL.
<b>Program continuation by</b>	Power ON
<b>26 003</b>	Axis %1 invalid lead screw pitch
<b>Explanation</b>	%1 = Axis name, spindle number The lead of the ball rolling/trapezoidal spindle set in the axis-specific MD 31 030 LEADSCREW_PITCH is zero.
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation. NC Start inhibited. NC Stop at alarm. The NC changes to follow-up mode. Alarm display. Interface signals are set.
<b>Remedy</b>	Determine the lead of the ball rolling spindle (information of the machine-manufacturer or measure the lead with the spindle cover removed) and enter the value in MD 31 030 LEADSCREW_PITCH (in most cases, 10 or 5 mm/rev).
<b>Program continuation by</b>	Power ON

<b>26 005</b>	Axis %1 invalid output rating configured
<b>Explanation</b>	%1 = Axis name, spindle number The output evaluation of the analog speed setpoint set in MD 32 260 RATED_VELO is illegal (e.g. zero).
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation. NC Start inhibited. NC Stop at alarm. The NC changes to follow-up mode. Alarm display. Interface signals are set.
<b>Remedy</b>	Enter the motor nominal speed in MD 32 260 RATED_VELO.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>26 006</b>	Axis %1 encoder %2 encoder type/output type %3 not possible
<b>Explanation</b>	%1 = Axis name, spindle number %2 = Encoder number %3 = Encoder type/output type Permissible settings: MD 30 240 ENC_TYPE           = 0     Simulation = 2     Square-wave encoder = 3     Step-switching motor selection MD 30 130 CTRLOUT_TYPE   = 0     Simulation = 1     Standard = 2     Step-switching motor selection
<b>Response</b>	Mode group ('BAG') not ready for operation. Channel not ready for operation. NC Start inhibited. NC Stop at alarm. The NC changes to follow-up mode. Alarm display. Interface signals are set.
<b>Remedy</b>	Check MD 32 240 ENC_TYPE and/or MD 30 130 CTRL_OUT_TYPE and correct if necessary.
<b>Program continuation by</b>	Control system OFF-ON.
<b>26 014</b>	Axis %1 machine data %2 invalid value
<b>Explanation</b>	%1 = Axis name, spindle number %2 = String: MD identifier MD contains invalid value.



---

<b>Response</b>	<p>Mode group ('BAG') not ready for operation.  Channel not ready for operation.  NC Start inhibited.  NC Stop at alarm.  The NC changes to follow-up mode.  Alarm display.  Interface signals are set.</p>
<b>Remedy</b>	Repeat your entry with the correct value and Power On.
<b>Program continuation by</b>	Power ON
<b>26 015</b>	Axis %1 machine data %2 [%3] invalid value
<b>Explanation</b>	<p>%1 = Axis name, spindle number  %2 = String: MD identifier  %3 = Index: MD array index  MD contains an invalid value.</p>
<b>Response</b>	<p>Mode group ('BAG') not ready for operation.  Channel not ready for operation.  NC Start inhibited.  NC Stop at alarm.  The NC changes to follow-up mode.  Alarm display.  Interface signals are set.</p>
<b>Remedy</b>	Repeat your entry with the correct value and Power On.
<b>Program continuation by</b>	Power ON

## 1.2 Cycle Alarms

<b>61 000</b>	No tool offset active
<b>Source (cycle)</b>	LCYC75
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Program a tool with tool offset in the called program.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 001</b>	Thread lead incorrectly defined
<b>Source (cycle)</b>	LCYC84 LCYC840
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Check parameter R106 (R106=0).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 002</b>	Type of machining incorrectly defined
<b>Source (cycle)</b>	LCYC93 LCYC95 LCYC97
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The value of parameters R105 for the machining type is incorrectly set and must be altered.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 003</b>	3rd. axis do not exist
<b>Source (cycle)</b>	LCYC82 LCYC83 LCYC84 LCYC840 LCYC85
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Check machine configuration and plane selection.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>61 004</b>	Configuration of axis is not correct
<b>Source (cycle)</b>	LCYC60 LCYC61 LCYC75
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Check machine configuration (one geometry axis missing).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 007</b>	First drilling depth incorrectly defined
<b>Source (cycle)</b>	LCYC83
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Alter the value for the first drilling depth (the first drilling depth is opposite to the total drilling depth).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 101</b>	Reference plane incorrectly defined
<b>Source (cycle)</b>	LCYC75 LCYC82 LCYC83 LCYC84 LCYC840 LCYC85
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Check parameters R101, R103, R104: R103=R104 or R103 is not between R101 and R104.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 102</b>	No spindle direction programmed
<b>Source (cycle)</b>	LCYC840
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Value of parameter R107 is greater than 4 or less than 3.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 103</b>	Number of holes is zero
<b>Source (cycle)</b>	LCYC60 LCYC61
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Parameter R119 = 0.

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 105</b>	Milling cutter radius too big
<b>Source (cycle)</b>	LCYC75
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The milling cutter radius in the tool offset memory is greater than the pocket or groove width (R119 or R118). Use smaller cutter or modify pocket width.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 106</b>	Number of or distance between circular elements
<b>Explanation</b>	LCYC61
<b>Response</b>	Block preparation in NC is aborted.
<b>Remedy</b>	Faulty parameterization of R119 or R1220. The arrangement of the circle elements on a full circle is not possible.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 107</b>	First drilling depth incorrectly defined
<b>Explanation</b>	LCYC83.
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Modify value for the first drilling depth (first drilling depth is opposite to the total drilling depth).
<b>Program continuation by</b>	Press the RESET key to clear the alarm.
<b>61 108</b>	Wrong milling direction
<b>Source (cycle)</b>	LCYC75
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Faulty parameter R126. This is greater than 3 or less than 2.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 109</b>	Parameterized cycle parameters incorrectly defined
<b>Explanation</b>	LCYC60 LCYC61
<b>Response</b>	Block preparation in NC is aborted.
<b>Remedy</b>	Parameter R115 is faulty. The parameterized cycle number is missing.

---

<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 601</b>	Finished part diameter too small
<b>Source (Cycle)</b>	LCYC94
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	A finished-part diameter < 3mm has been programmed. Increase value.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 602</b>	Tool width incorrectly defined
<b>Source (Cycle)</b>	LCYC93
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The tool width (parameter R107) does not match with the programmed recess shape.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 603</b>	Recess type incorrectly defined
<b>Source (Cycle)</b>	LCYC93
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The recess form is incorrectly programmed.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 605</b>	Contour incorrectly defined
<b>Explanation</b>	LCYC95
<b>Response</b>	Block preparation in NC is aborted.
<b>Remedy</b>	Contour contains undercut elements that must be removed.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 606</b>	Error during contour preparation
<b>Explanation</b>	LCYC95
<b>Response</b>	Block preparation in NC is aborted.
<b>Remedy</b>	Check contour subroutine. Check machining parameter (R105).
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

<b>61 607</b>	Starting point incorrectly defined
<b>Source (Cycle)</b>	LCYC95
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The start point reached prior to the cycle call is not outside the square described by the contour subroutine.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 608</b>	Indirect tool point direction programmed
<b>Source (Cycle)</b>	LCYC94
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	An edge position 1 .. 4 matching with the recess shape must be programmed.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 609</b>	Shape incorrectly defined
<b>Source (Cycle)</b>	LCYC94
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	Check parameters for the recess shape.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>61 610</b>	Infeed depth not programmed
<b>Source (Cycle)</b>	LCYC95
<b>Response</b>	Block preparation in NC is aborted
<b>Remedy</b>	The parameter R108>0 for the infeed depth must be programmed for roughing.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.
<b>62 000</b>	Channel %1 block %2
<b>Explanation</b>	%1 = Channel number %2 = Block number, label %3 = Text index for cycle alarms See SIEMENS Cycle Description
<b>Response</b>	Alarm display.
<b>Remedy</b>	See SIEMENS Cycle Description
<b>Program continuation by</b>	Press the Clear key to clear the alarm. No further operation required.

---

<b>62 101</b>	Finishing allowance has been reduced to possible value
<b>Explanation</b>	LCYC75
<b>Response</b>	Block preparation in NC is aborted.
<b>Remedy</b>	The cutter diameter is larger than the pocket or groove width minus the finishing allowance. Either use a smaller cutter, or reduce the finishing allowance, or enlarge the pocket/groove width.
<b>Program continuation by</b>	Press the RESET key to clear the alarm. Restart part program.

## 1.3 PLC Alarms

<b>400 000</b>	PLC Stop [type]
<b>Explanation</b>	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)
<b>Response</b>	Alarm display
<b>Remedy</b>	Eliminate other PLC alarm; set start-up switch to a position other than 4, or test user program.
<b>Program continuation by</b>	POWER ON Select Start-Up menu to continue Start via PT 802 Programming Tool
<b>400 002</b>	System error [type]
<b>Explanation</b>	This alarm refers you to internal error states that in conjunction with the error number displayed provide you with information on error cause and error location.
<b>Response</b>	PLC stop
<b>Remedy</b>	Please report this error with type number to Siemens. For error cause refer to the specified software section in the displayed line number
<b>Program continuation by</b>	POWER ON
<b>400 003</b>	Internal connection faulty
<b>Explanation</b>	An internal fault of connection between operator panel and PLC has been detected during cyclic operation.
<b>Response</b>	Emergency Stop
<b>Remedy</b>	Contact SIEMENS hotline for repair.
<b>Program continuation by</b>	Press CLEAR to clear the alarm.
<b>400 004</b>	Code error: Network [No.] [string]
<b>Explanation</b>	[ No. ] = Network number [ string ] = internal error code The user program contains an operation which is not supported by the control system.
<b>Response</b>	PLC stop
<b>Remedy</b>	Modify user program and reload.
<b>Program continuation by</b>	POWER ON



<b>400 005</b>	Start-up switch set to PLC stop
<b>Explanation</b>	The user program is not executed
<b>Response</b>	Alarm display
<b>Remedy</b>	Set the start-up switch to a position other than 4.
<b>Program continuation by</b>	The alarm display disappears with the alarm cause.
<b>400 006</b>	Buffered PLC data deleted
<b>Explanation</b>	This can have the following causes: <ul style="list-style-type: none"> <li>• Operator action (e.g. PLC overall reset, boot with default values)</li> <li>• Operator action "Boot with saved data", without first saving data</li> <li>• Back-up time exceeded</li> </ul>
<b>Response</b>	Alarm display
<b>Remedy</b>	Update required data.
<b>Program continuation by</b>	Press the Clear key to clear the alarm.
<b>400 007</b>	Operand error: Network [No.] [string]
<b>Explanation</b>	%1 = Network number %2 = Faulty variable
<b>Response</b>	PLC stop
<b>Remedy</b>	Check the displayed variable in the user program for address range violation, invalid data type and alignment errors.  If the error concerns PLC inputs or outputs, missing or defective I/O modules may be the cause.
<b>Program continuation by</b>	POWER ON
<b>400 008</b>	Programmed tool version not compatible [version]
<b>Explanation</b>	This version is not compatible with the product stage of the control system.
<b>Response</b>	PLC stop
<b>Remedy</b>	Compile the user program with a compatible Programming Tool version and load it into the control system.
<b>Program continuation by</b>	POWER ON
<b>400 009</b>	Runtime overflow at PLC in network
<b>Explanation</b>	Check the user program of the respectively displayed network.
<b>Response</b>	PLC stop
<b>Remedy</b>	Modify user program.
<b>Program continuation by</b>	POWER ON

**400 010** Arithmetic error in network of user program: [No.] [type]  
**Explanation** Check the user program in the specified network.  
 [No.] Network number  
 Type 1: Division by zero with normal arithmetic  
 Type 2: Floating point error  
**Response** PLC stop  
**Remedy** Modify user program.  
**Program continuation by** POWER ON

**400 011** Permitted number of subroutine levels in network exceeded [No.]  
**Explanation** [No.] Network number  
 Check the user program in the specified network.  
**Response** PLC stop  
**Remedy** Modify user program.  
**Program continuation by** POWER ON

**400 013** PLC user program faulty  
**Explanation** The PLC user program in the control system is defective or missing.  
**Response** PLC stop  
**Remedy** Reload PLC user program.  
**Program continuation by** POWER ON

## 1.4 Action List

No./Name	Explanation	Not allowed if	Remedy
1. INIT	Carry out INI phase (the tasks are initialized after Power On)		
2. RESET	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	Changing from operating mode to program mode MDA or Automatic (VDI signal: BAG)	<ol style="list-style-type: none"> <li>1. the channel is active (program running, block search, loading of machine data)</li> <li>2. the program has already been started in another program mode.</li> <li>3. a channel has left the mode group because of an interrupt.</li> <li>4. Overstore or Digitalize has been selected.</li> </ol>	<p>⇒ Press RESET to abort the program or stop program (not during block search or machine data loading)</p> <p>⇒ Abort program using the Reset key</p> <p>⇒ Press RESET to abort the program or wait until interrupt is completed.</p> <p>⇒ Deselect Overstore, Digitalize</p>
6. MODESWITCHTOSAV E-MODE	Automatic change from an internal operating mode to the externally set operating mode (During TEACH_IN, it is tried after each stop to switch over from the internal operating mode "AUTOMATIC, MDA" to TEACH_IN)		

No./Name	Explanation	Not allowed if	Remedy
7. MODESWITCHTOHAND-MODE	Changing from the operating mode to a manual mode (VDI signal: JOG, TEACH_IN, REF)	<ol style="list-style-type: none"> <li>1. too high nesting depth: 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.</li> <li>2. the channel is active (program running, block search, loading of machine data)</li> <li>3. a channel has left the mode group because of an interrupt.</li> <li>4. Overstore or Digitalize is selected.</li> </ol>	<p>⇒ Press RESET to abort the program</p> <p>⇒ Press RESET to abort the program or stop the program (not during block search or machine data loading)</p> <p>⇒ Press RESET to abort the program or wait until the interrupt is completed.</p> <p>⇒ Deselect Overstore / Digitalize</p>
8. OVERSTOREON	Selection of Overstore mode (PI command).		
9. OVERSTOREOFF	Deselection of Overstore mode (PI command).		
10. SYNCHWITHREPOS	Synchronize advance and main part of program, e.g. switch on open-loop axis control (VDI signal: follow-up mode)		
11. INTERRUPT	Carry out user interrupt "ASUP" (VDI signal: digital-analog interface, ASUP interface).	<ol style="list-style-type: none"> <li>1. the channel is active due to block search or machine data loading</li> <li>2. the channel is stopped and the Asup "ASUP_START_MASK" must be started and the current block cannot be reorganized.</li> <li>3. Digitalize is selected</li> <li>4. reference-point approach has not yet been carried out</li> <li>5. the current block in which program execution has been stopped cannot be reorganized, since deceleration is carried out after several blocks.</li> </ol>	<p>⇒ Wait until block search or machine data loading is completed, or press the RESET key to abort the program</p> <p>⇒ Activate block change as long as the NC block can be reorganized.</p> <p>⇒ Deselect Digitalize</p> <p>⇒ Carry out reference-point approach or ignore this condition by setting the machine data "ASUP_START_MASK".</p> <p>⇒ Abort program</p>
12. INTERRUPTFASTLIFT-OFF	Carry out user interrupt "ASUP" with quick lifting (VDI signal: digital-analog interface)	See 11	

No./Name	Explanation	Not allowed if	Remedy
13. INTERRUPTBLSYNC	Carry out user interrupt at end of block (VDI signal: digital-analog interface)	See 11	
14. FASTLIFTOFF	Carry out quick lifting (VDI signal: digital-analog interface and ASUP interface, further actions are: 11, 12, 13, 88, 89).		
15. TM_MOVETOOL	Move tool - with tool management only - (PI command)		
16. DELDISTOGO	Carry out "Delete distance to go" (VDI signal: Delete distance to go, synchronous response)	1. too high nesting depth 2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Abort program ⇒ Abort program
17. FASTDELDISTOGO	Carry out "Delete distance to go quickly". Block execution is not reorganized (VDI signal: Delete distance to go, synchronous response)		
18. DELDISTOGOWITHO UT_STOP	Carry out "Delete distance to go quickly". Reorganization and stop movement are not carried out. (VDI signal: Delete distance to go, synchronous response)		
19. PROGRESETREPEAT	Abort subroutine repetition (VDI signal: Delete number of subroutine passes)	1. too high nesting depth 2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Abort program ⇒ Abort program
20. PROGCANCELSUB	Abort subroutine execution (VDI signal: program level abortion)	1. too high nesting depth 2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Abort program ⇒ Abort program
21. SINGLEBLOCKSTOP	Activate single block (VDI signal: Activate single block)		
22. SINGLEBLOCKOFF	Switch off single block. (VDI signal: Activate single block)		
23. SINGLEBLOCK_IPO	Activate main run single block (BTSS variable and VDI signal: Activate single block)		

## Alarms

No./Name	Explanation	Not allowed if	Remedy
24. SINGLEBLOCK_DECODIER	Activate decoding single block (BTSS variable and VDI signal: Activate single block)	1. too high nesting depth  2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Wait until the preceding Asup is completed, or abort program ⇒ Abort program
25. SINGLEBLOCK_MAINBLOCK	Activate main program single block (BTSS variable and VDI signal: Activate single block)		
26. SINGLEBLOCK_PATH	Activate traversing single block (BTSS variable and VDI signal: Activate single block)		
27. 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
28. 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 (Automatic only)	⇒ Secure Start with WAITE ⇒ Carry out alarm clear condition ⇒ Approach reference point ⇒ Select program mode
29. 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
30. 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
31. 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
32. STOPALL	Stop all axes. (VDI signal: Stop All or Reset key)		
33. STOPPROG	Carry out program stop. (NC block: M0)		
34. STOPJOGREF	Stop JOG movement (VDI signal: NC Stop)		
35. STOPDIGITIZE	Stop digitalization. (VDI signal: NC Stop)		

No./Name	Explanation	Not allowed if	Remedy
36. STARTSIG	Start selected program mode (VDI signal: NC Start)	1. Process switch active (mode change, Digitalize ON/OFF, Overstore ON/OFF) 2. an alarm response is provided which prevent start or forces deceleration. 3. a process is running (NC program, block search, loading of machine data)	⇒ -  ⇒ Carry out alarm clear condition  ⇒ -
37. STOPSIG	Stop active program mode (VDI signal: NC Stop)		
38. INITIALINISTART	Start machine data processing (INI file is already in NCK), (PI command)		
39. INITIALINIEXTSTART	Start machine data processing (INI file is externally, e.g. on MMC), (PI command)		
40. BAGSTOP_SLBTYPA	Stop due to single block mode. VDI signal, individual type A (only executable blocks), after stop in other channel of this mode group		
41. BAGSTOPATEND_SLBTYPB	Stop due to single block mode group. VDI signal, individual type A (any blocks), after stop at end of block in other channel of this mode group		
42. OVERSTORE_BUFFER_END_REACHED	Stop because end of overstore buffer “_N_OSTOREXX_SYF” has been reached.		
43. PREP_STOP	Start block search (NC block: Stopre)		
44. PROG_STOP	Stop program execution at end of block (NC block: M00/M01)		
45. STOPPROGABLOCK END	Stop program execution at end of block (alarm, VDI signal: NC stop at end of block)		
46. STOPPROGATASUP END	Stop at ASUP end, is start has been carried out from “Stopped”.		
47. PROGSELECT	Select program. (PI command)		
48. PROGSELECTTEXT	Select the program which is already on an external (PI command)		
49. CHANNEL_PROG SELECT	Program selection from another channel (channel communication, NC block: INIT)		

## Alarms

No./Name	Explanation	Not allowed if	Remedy
50. ASUPDEFINITION	Save definition of ASUPS which can be activated (PI command)		
51. NEWCONF	Sets all machine data with (NEW_CONF) attribute to active state (PI command)		
52. CLEARCANCELALARM	Delete all alarms with CANCELCLEAR clear condition (PI command, Quit Alarm key)		
53. BLOCKSEARCHUN_CONTINUE	Continue block search. (NC block, Stopre)		
54. BLOCKSEARCHRUN_START	Start block search. (PI command)		
55. BLOCKSEARCHRUN_RESUME	Continue block search (PI command)		
56. DIGITIZEON	Activate digitalization (PI command)		
57. DIGITIZEOFF	Deactivate digitalization (PI command)		
58. FUNCTGENON	Switch on function generator (PI command)		
59. FUNCTGENOFF	Switch off function generator (PI command)		
60. WAITM	Wait for program marker (channel communication, NC block: WAITM)		
61. WAITE	Wait for end of program (channel communication, NC block: WAITE)		
62. INIT_SYNC	Program selection from another channel, synchronization (Channel communication, NC block: INIT)		
63. MMC_CMD	Wait until acknowledgment from MMC (NC block, MMC_CMD)		
64. PROGMODESLASHON	Activate block skip of blocks that can be skipped (VDI signal: Skip block)	too high nesting depth	⇒ Wait until the preceding ASUP is completed, or abort program
65. PROGMODESLASH OFF	Deactivate block skip of blocks that can be skipped (VDI signal: Skip block)	too high nesting depth	⇒ Wait until the preceding Asup is completed, or abort program
66. PROGMODEDRYRUN ON	Activate test run (VDI signal: Rapid traverse superimposition)	1. too high nesting depth 2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Wait until the preceding ASUP is completed, or abort program ⇒ Abort program



No./Name	Explanation	Not allowed if	Remedy
67. PROGMODEDRYRUN OFF	Deactivate test run (VDI signal: Rapid traverse superimposition I)	1. too high nesting depth  2. the current block in which program execution is stopped cannot be reorganized, since deceleration is carried out over several blocks.	⇒ Wait until the preceding ASUP is completed, or abort program ⇒ Abort program
68. BLOCKREADINHIBIT_ ON	Activate read-in disable for main run block (VDI signal: Read-in inhibit)		
69. BLOCKREADINHIBIT_ OFF	Deactivate read-in disable for main run block (VDI signal: Read-in inhibit)		
70. STOPATEND_ALARM	Stop at end of block (alarm)		
71. STOP_ALARM	Stop all axes (alarm)		
72. 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 abort program or process, or wait for end of program
73. PROGTESTOFF	Deactivate program test (VDI signal: Program test)	NCK channel condition is not READY	⇒ Press RESET to abort program or process, or wait for end of program
74. STOPATIPOBUFFER_ ISEMPTY_ALARM	Stop at end of block preparation (alarm)		
75. STOPATIPOBUF_ EMPTY_ALARM_ REORG	Stop at end of block preparation with following reorganization of block execution (alarm)	nesting depth too high	⇒ Wait until preceding Asup is completed, or abort program
76. CONDITIONAL_STOP ATEND	Conditioned stop at end of block (If after continuation by NC Start a reason for stop "Stop at end of block" is still provided, Stop is carried out again.)		
77. CONDITIONAL_SBL_ DEC_STOPATEND	Conditioned stop at end of block (Despite of start, no block is moved by the interpreter or advance to the main run.)		
78. INTERPRETERSTOP_ ALARM	Stop block search (alarm)		
79. RETREAT_MOVE_ THREAD	Retraction movement in the case of G33 and stop.		
80. WAITMC	Conditioned waiting for program marker (NC block: WAITMC)		
81. SETM	Set marker (NC block: SETM)		
82. CLEARM	Delete marker (NC block: CLEARM)		
83. BLOCK_SELECT	Select NC block (PI command)		

Alarms

No./Name	Explanation	Not allowed if	Remedy
84. LOCK_FOR_EDIT	Block currently executed NC program for editing (PI command)		
85. START_TEACHINPROG	Start program in TEACHIN submode (VDI signal: NC Start)	see 36 and 5	
86. RESUME_TEACHINPROG	Continue program in TEACHIN submode (VDI signal: NC-Start)	see 36 and 5	
87. PURE_REORG	Reorganize block execution.		
88. INTERRUPT_TOPROG_NOREPOS	Activate user interrupt "ASUP" in a manual mode (VDI signal: ASUP, digital-analog interface)	see 11	
89. INTERRUPT_START	Activate user interrupt "ASUP"; is only carried out in channel condition READY (VDI signal: ASUP, digital-analog interface)	see 11	
90. INTERRUPT_SIGNAL	Carry out user interrupt "ASUP" (VDI signal: ASUP, digital-analog interface, further actions are: 11, 12, 13, 88, 89)	see 11	
91. STOPBAG	Stop program execution (VDI signal: BAG Stop)		
92. NEWCONF_PREP_STOP	Set all machine data with NEW_CONF attribute to active condition (NC_block: NEW_CONF)		
93. BLOCKSEARCHRUN_NEWCONF	Set all machine data with NEW_CONF attribute to active condition (NC block: NEW_CONF during block search)		
94. CONTINUE_INTERPR	Start continuation of interpreter processing (internal block search stop)		

# Abbreviations / Glossary

# 2

## 2.1 Abbreviations

<b>A</b>	Output
<b>ASCII</b>	American Standard Code for Information Interchange
<b>AV</b>	Preparation for work
<b>BA</b>	Operating mode
<b>BAG</b>	Operating mode groups
<b>BB</b>	Ready for operation
<b>BCD</b>	Binary Coded Decimals
<b>BHG</b>	Hand-held terminal
<b>BOF</b>	User interface
<b>CNC</b>	Computerized Numerical Control
<b>CP</b>	Communication Processor
<b>CPU</b>	Central Processing Unit
<b>CR</b>	Carriage Return
<b>CSB</b>	Central Service Board (PLC module)
<b>CTS</b>	Clear To Send
<b>DAU</b>	Digital-Analog Converter
<b>DB</b>	Data Block
<b>DIN</b>	German Industrial Standards
<b>DIO</b>	Data Input/Output
<b>DRF</b>	Differential Resolver Function
<b>DRY</b>	Dry Run

<b>DSB</b>	Decoding Single Block
<b>DSR</b>	Data Send Ready
<b>DW</b>	Data Word
<b>E</b>	Input
<b>EIA-Code</b>	Special tape code, number of holes per characters always odd
<b>EPROM</b>	Program memory with fixed program
<b>E/R</b>	Controlled Supply and Energy Recovery Module
<b>ETC</b>	ETC key: Extension of the softkey bar in the same menu
<b>FDB</b>	Product designation database
<b>FIFO</b>	First in First Out
<b>FRA</b>	Frame module
<b>FRAME</b>	Coordinate conversion with the components zero offset, rotation, scaling, mirror-imaging
<b>FRK</b>	Cutter radius compensation
<b>FST</b>	Feed Stop
<b>GUD</b>	Global User Data
<b>HMS</b>	High-Resolution Measuring System
<b>HSA</b>	Main Spindle Drive
<b>HW</b>	Hardware
<b>IM</b>	Interface Module
<b>IM-S/R</b>	Interface Module (S=send/R=receive)
<b>INC</b>	Increment
<b>ISO-Code</b>	Special tape code, number of holes per character always even
<b>K1...K4</b>	Channel 1 to Channel 4
<b>KOP</b>	Ladder Diagram
<b>K<sub>v</sub></b>	Loop-Gain Factor
<b>K<sub>ü</sub></b>	Transformation Ratio
<b>LCD</b>	Liquid Crystal Display

<b>LED</b>	Light Emitting Diode
<b>LUD</b>	Local User Data
<b>MB</b>	Megabyte
<b>MD</b>	Machine Data
<b>MK</b>	Measuring Circuit
<b>MDA</b>	Manual Data Automatic
<b>MLFB</b>	Machine-readable product designation
<b>MPF</b>	Main Program File: NC part program (main program)
<b>MPI</b>	Multi Point Interface
<b>MSTT</b>	Machine control panel
<b>NC</b>	Numerical Control
<b>NCK</b>	Numerical Control Kernel (numerical kernel with block preparation, traversing range etc.)
<b>NCU</b>	Numerical Control Unit
<b>NURBS</b>	Non Uniform Rational B Spline
<b>NV</b>	Zero Offset
<b>OEM</b>	Original Equipment Manufacturer
<b>OP</b>	Operator Panel
<b>OPI</b>	Operator Panel Interface
<b>PC</b>	Personal Computer
<b>PCMCIA</b>	Personal Computer Memory Card International Association:
<b>PG</b>	Programming Device
<b>PLC</b>	Programmable Logic Control
<b>PRT</b>	Program Test
<b>RAM</b>	Random Access Memory
<b>RISC</b>	Reduced Instruction Set Computer
<b>ROV</b>	Rapid Override
<b>RPA</b>	R Parameters Active (NCK memory area for R parameter numbers)

<b>RTS</b>	Request To Send (control signal from serial data interfaces)
<b>SBL</b>	Single Block
<b>SBL2</b>	Single Block Decoding
<b>SEA</b>	Setting Data Active (memory area for setting data in NCK)
<b>SD</b>	Setting Data
<b>SKP</b>	Skip Block
<b>SM</b>	Signal Module
<b>SPF</b>	Sub Program File
<b>SPS</b>	Programmable Logic Controller
<b>SRK</b>	Cutter Radius Compensation
<b>SSFK</b>	Leadscrew Error Compensation
<b>SSI</b>	Serial Synchronous Interface
<b>SW</b>	Software
<b>TEA</b>	Testing Data Active (with reference to the machine data)
<b>TO</b>	Tool Offset
<b>TOA</b>	Tool Offset Active (memory area for tool offsets)
<b>TRANSMIT</b>	Transform Milling into Turning (coordinate conversion on turning machines for milling)
<b>VSA</b>	Feed drive (spindle)
<b>V</b>	Bit type PLC variable
<b>VB</b>	Byte type PLC variable
<b>WKZ</b>	Tool
<b>WZ</b>	Tool
<b>WZK</b>	Tool offset
<b>ZOA</b>	Zero Offset Active (memory area for zero offsets)

## 2.2 Glossary

<b>User program</b>	Total of all status bar graphs in an executable form in the PLC
<b>Command</b>	Instruction in user program
<b>Operating mode</b>	Mode of program execution, e.g. manual mode, automatic mode, corresponding to the machine operation
<b>Diagnosis</b>	Detection of faulty processes during program execution; locates undesired or unexpected phenomena in the process
<b>Multi-point interface</b>	MPI Hardware module for online coupling with the user program
<b>Sensor</b>	Electrical element; provides a signal to the control system





SIEMENS AG  
A&D MC BMS  
Postfach 3180  
D-91050 Erlangen

(Tel. +49 180 / 5050 – 222 [Hotline]  
Fax +49 9131 / 98 – 2176 [Documentation]  
Mailto: motioncontrol.docu@erlf.siemens.de)

<b>From</b>	<b>Suggestions</b>
Name _____	<b>Corrections</b>
Company/dept. _____	for Publication/Manual:
Street _____	SINUMERIK 802S/802C base line
Zip code: _____ City: _____	User Documentation
Telephone: _____ / _____	Diagnostics Guide
Telefax: _____ / _____	Order No.: 6FC5598-4AA21-0BP0 Edition: 08.03
	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**



---

**SINUMERIK 802S/C base line Document Structure**

---

General Documentation: **Catalog**

Turning  
Milling

User Manual: **Operation and Programming**

Short  
Guide for  
O&P

Turning

Milling

User Manual: **Diagnostics Guide**

Turning  
Milling

Technical Manual: **Start-Up**

Quick  
Start

802S  
base line  
Start-Up

802C  
base line  
Start-Up

Technical Manual: **Description of Functions**

Turning  
Milling

**Siemens AG**

Automatisierungs- und Antriebstechnik

Motion Control Systems

Postfach 3180, D – 91050 Erlangen

Germany

[www.ad.siemens.de](http://www.ad.siemens.de)

© Siemens AG 2003  
Subject to change without prior notice  
Order No.: 6F5598-4AA21-0BP0

Printed in the Federal Republic of Germany