SIEMENS

SINUMERIK

SINUMERIK 808D
Diagnostics Manual

Diagnostics Manual

Valid for:
SINUMERIK 808D Turning (software version: V4.4.2)
SINUMERIK 808D Milling (software version: V4.4.2)

Target group:
Project engineers, commissioning engineers, machine
 operators, and service and maintenance personnel

Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

> ### ⚠ DANGER
> indicates that death or severe personal injury **will** result if proper precautions are not taken.

> ### ⚠ WARNING
> indicates that death or severe personal injury **may** result if proper precautions are not taken.

> ### ⚠ CAUTION
> indicates that minor personal injury can result if proper precautions are not taken.

> ### NOTICE
> indicates that property damage can result if proper precautions are not taken.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

> ### ⚠ WARNING
> Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Table of contents

Preface

SINUMERIK 808D documentation

The SINUMERIK 808D documentation consist of the following components:

- Operating Instructions
    - Mechanical Installation Manual
    - Electrical Installation Manual
    - PLC Subroutines Manual
    - Function Manual
    - Parameter Manual
- Diagnostics Manual
- Commissioning Manual
- Programming and Operating Manual (Turning)
- Programming and Operating Manual (Milling)
- Manual Machine Plus (Turning)
- Online Help for Programming and Operating (Turning)
- Online Help for Programming and Operating (Milling)
- Online Help for Manual Machine Plus (Turning)

My Documentation Manager (MDM)

Under the following link you will find information to individually compile your documentation based on the Siemens content:

www.siemens.com/mdm

Target group

This manual is intended for use by project engineers, commissioning engineers, machine operators and service and maintenance personnel.

Benefits

This manual enables the intended target groups to evaluate error and fault indications and to respond accordingly.

It allows the operator at the machine tool:

- To correctly assess special situations when operating the machine.
- To ascertain the reaction of the system to the special situation.

- To utilize the possibilities for continued operation following the special situation.
- To follow references to other documentation containing further details.

Technical support

Hotline:	+86 400-810-4288
Service and Support	• China: www.siemens.com.cn/808D • Worldwide: http://support.automation.siemens.com

EC Declaration of Conformity

The EC Declaration of Conformity for the EMC Directive can be found on the Internet at http://support.automation.siemens.com

Here, enter the number **15257461** as the search term or contact your local Siemens office.

Licensing provisions

The SINUMERIK 808D software is protected by national and international copyright laws and agreements. Unauthorized reproduction and distribution of this software or parts thereof is liable to prosecution. It will be prosecuted both according to criminal and civil law and may result in severe penalties or claims for compensation.

In the SINUMERIK 808D, open source software is used. The licensing provisions for this software are included on the Toolbox DVD and are to be observed accordingly.

Introduction

1

1.1 Structure of the Diagnostics Manual

NCK / PLC alarms

The descriptions for the alarms can be found in the chapters:

- NCK alarms (Page 49)
- PLC alarms (Page 289)
- SINAMICS V60 alarms (Page 297)

In each chapter, the alarm descriptions are sorted according to ascending alarm number. There are gaps in the sequence.

Structure of the NCK / PLC alarm descriptions

The descriptions of the alarms have the following layout:

\<Alarm No.\>   **\<Alarm text\>**

Explanation:

Reaction:

Help:

Continue program:

Each alarm is uniquely identified using the \<Alarm number\> and the \<Alarm text\>.

The description of the alarms is classified according to the following categories:

- Explanation
- Response

  See Chapter: System reactions to SINUMERIK alarms (Page 293)
- Remedy
- Program continuation

  See Chapter: System reactions to SINUMERIK alarms (Page 293)

Action list

The actions described in the NCK alarm texts ("Action %...") are explained in the following Chapter:

See Chapter: Cancel criteria for alarms (Page 296)

Specification "%"

The specification "%" represents variables for an online parameter that is replaced on the control with a corresponding value. Since 808D is a single-channel control system, "Channel 1" is always output.

Safety

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

1.2   Alarm number ranges

The following tables show an overview of all of the reserved number ranges for alarms / messages.

Note
In the alarm lists of this *Diagnostics Manual*, only those number ranges are represented, which are valid for the specified product.

Table 1-1   NCK alarms / messages

002 000 - 009 999	General alarms
010 000 - 019 999	Channel alarms
020 000 - 029 999	Axis / spindle alarms
030 000 - 099 999	Functional alarms

NCK alarms (Page 49)

Table 1-2   PLC alarms / messages

400 000 - 499 999	General PLC alarms

PLC alarms (Page 289)

Operating in the "SYSTEM" area

<div style="text-align: right; font-size: 3em;">2</div>

2.1 "SYSTEM" operating area

Overview

The "SYSTEM" operating area includes functions required for parameterizing and analyzing the NCK, the PLC and the drive. In this chapter, all functions in the system operating area are described except for the "Save data" window which is described in Chapter "Data backup (Page 303)".

System screen layout

Press the [SHIFT] and [SYSTEM ALARM] keys on the PPU to enter the "SYSTEM" operating area. The start screen displays the machine configuration data and softkeys available. Depending on the functions selected, the horizontal and the vertical softkey bars vary. The screenshot below uses the turning variant as an example.
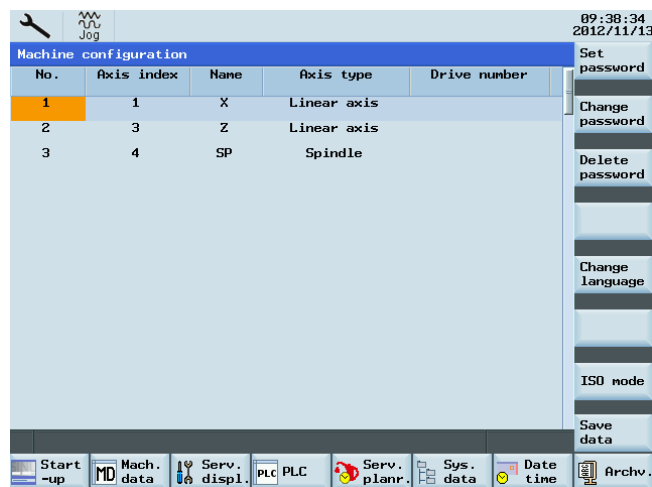
Softkeys

The following table shows the explanations about softkeys on the main screen.

Softkey name	Function
Start-up	Sets the NC, PLC and HMI start up modes.
Mach. data	Sets the system machine data.
Serv. displ.	Views the service information.
PLC	Provides PLC commissioning and diagnostics.
Serv. planr.	Defines the maintenance planner.

Softkey name	Function
Sys.data	Backs up system data.
Date time	Sets the data and time shown on the screen.
Archv.	Creates and restores start-up or series start-up archives.
Set password	Enters the corresponding password (system password, manufacturer password, and end user password) to access the different user levels.
Change password	Changes the password as per the corresponding access levels.
Delete password	Deletes the current password.
Change language	Selects the user interface language. Note that the HMI is automatically restarted when a new language is selected.
ISO mode	Switches to the ISO programming mode.
Save data	Saves the content of the volatile memory into the non-volatile memory.

2.2 Setting start-up function

Functionality

The "Start-up" softkey allows you to choose the NC and PLC start up modes.

You can set up the NC or PLC start up modes by pressing the respective softkeys.

Selecting the NC restart mode

To select the NC restart mode, follow these steps:

1. Press the "Start-up" softkey on the main screen of the "SYSTEM" operating area.
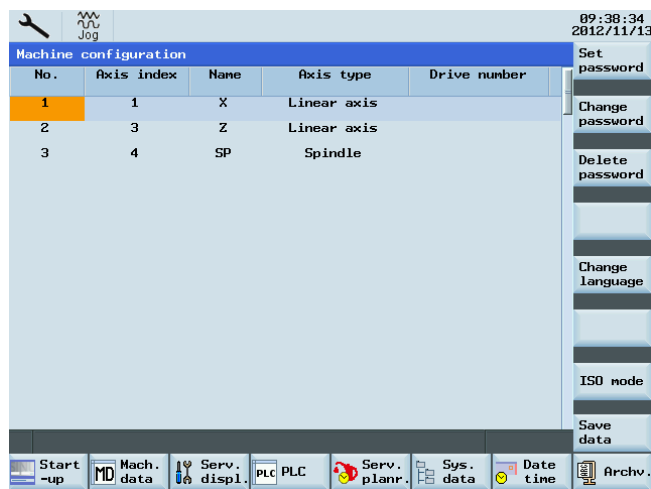
2. By default, the NC restart mode setting screen opens. Select the desired mode with the cursor keys. Three NC restart modes are available: Standard power-up, Power-up with default data, and Power-up with saved data.

3. Press the "OK" softkey or the ∧ key on the PPU to confirm or cancel your selection.

4. If you press the "OK" softkey, the system then restarts in the mode selected.
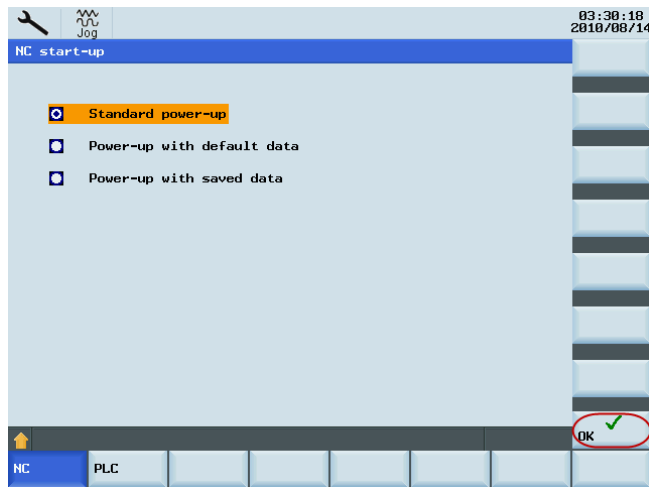
Note:

- You must stop the machine before selecting the NC start mode.

- When you choose "Standard power-up", the system is restarted immediately after your confirmation; for "Power-up with default data", the system is restarted with the default values which were supplied at the initial state; for "Power-up with saved data", the system is restarted with the data that was last saved.

Selecting the PLC restart mode

To select the PLC restart mode, follow these steps:

1. Press the "Start-up" softkey on the main screen of the "SYSTEM" operating area, and then press the "PLC" softkey. The PLC start-up window opens. Three PLC restart modes are available: Restart, Overall reset and Debug mode.

2. Select "Restart" or "Overall reset" with the cursor keys.

3. Press the <SELECT> key to select "Debug mode".

4. Press the "OK" softkey or the $\land$ key on the PPU to confirm or cancel your selection.

5. If you press the "OK" softkey, the system then restarts in the mode selected.
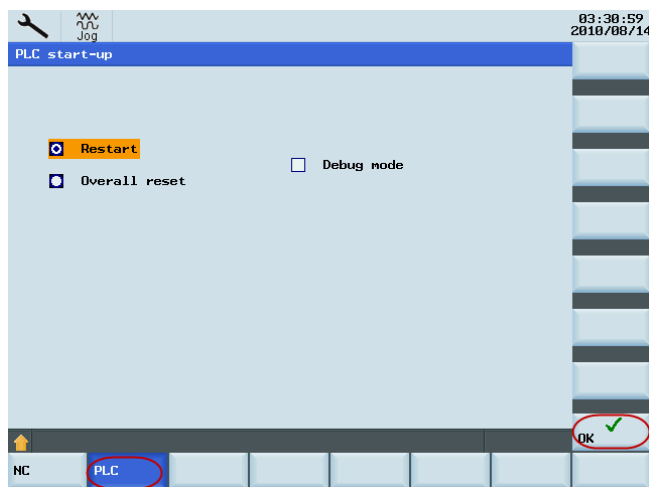
2.3 Setting system machine data

References

You can find a description of the machine data in the following manufacturers´ documents:

SINUMERIK 808D Parameter Manual

SINUMERIK 808D Function Manual

Setting basic data

1. Press the "Mach.data" softkey on the main screen of the "SYSTEM" operating area.
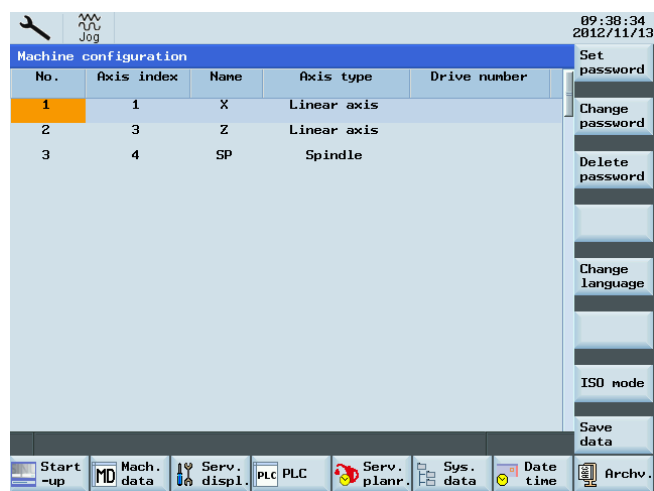
2. By default, the "Basic machine data" screen opens. Enter the values in the desired blanks and press the <INPUT> key to confirm.
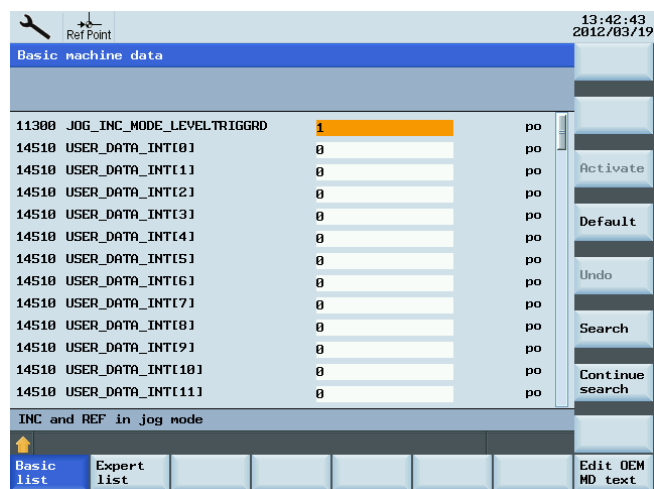
SINUMERIK 808D provides an easy-to-use data list for beginner users. In the basic data list, the general, axis, and channel MD are integrated in one screen.

The following describes the machine data structure in detail:

Figure 2-1   Structure of a machine data line

Table 2-1   Legend

No.	Meaning		
1	MD number		
2	Name		
3	Value		
4	Unit		
5	Effective	im	immediately effective
		cf	with confirmation
		re	Reset
		po	Power on

⚠ CAUTION

Machine damage

Incorrect parameterization may result in destruction of the machine!

Setting General MD

1. Press the "Expert List" softkey to set the General, Axis, Channel, and Display MD.

   In the expert data list, you can set the general, axis, channel, and display MD by pressing the respective extension softkeys.

2. By default, the "General machine data" window opens. Use the <PAGE DOWN>/<PAGE UP> key to browse forward/backward.

3. Enter the values in the desired blanks and press the <INPUT> key to confirm. Any changes in the machine data have a substantial influence on the machine.

Softkeys

The following table shows the description of softkeys in detail:

Softkey name	Function
Default	Undoes the modified values and recalls the default values.
Search	Searches for the desired number or the name (or a part of the name) of the machine data.
Continue search	Continues searching for the next match
Select group	Provides various display filters for the active machine data group
Activate	Activates the modified values.
Undo	Undoes the modified values and recalls the lastly used values.

Setting Axis-specific machine data

To set the axis-specific machine data, follow these steps:

1. Press the "Axis MD" softkey. The "Axis-specific machine data" window opens. The softkeys "Axis +" and "Axis - -" appear on the vertical softkey bar.

2. Enter the values in the desired blanks and press the <INPUT> key to confirm.

3. Use the softkey "Axis +" or "Axis - -" to switch to the machine area of the next or previous axis.
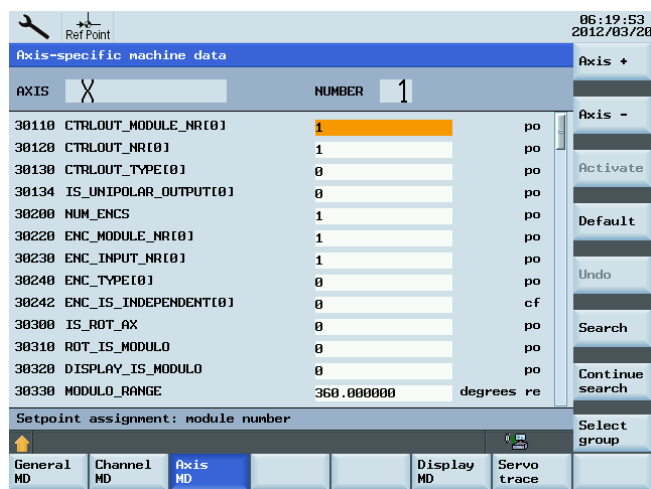
Setting Channel-specific machine data

1. Press the "Channel MD" softkey. The "Channel-specific machine data" window opens.

2. Enter the values in the desired blanks and press the <INPUT> key to confirm.

3. Use the cursor keys to browse forward / backward.

Displaying machine data

1. Press the "Display MD" softkey. The "Display machine data" window opens.

2. Enter the values in the desired blanks and press the <INPUT> key to confirm.

3. Use the cursor keys to browse forward / backward.

Checking MD by servo trace

By pressing the "Servo trace" softkey, you can compare the defined machine data with the actual data and adjust the parameters if necessary.

To learn more functions regarding the servo trace, refer to the Section "Servo trace (Page 22)".

2.4   Viewing the service info

Service display

To view the service display, follow these steps:

1. Select the "Serv. displ." softkey on the "SYSTEM" start screen.
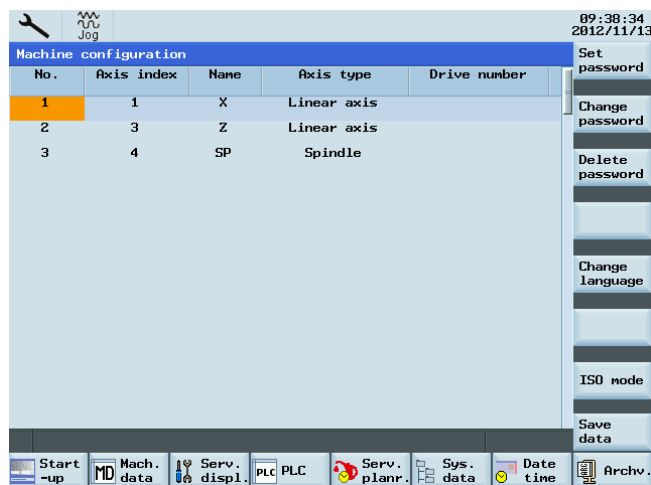
2. The screen for the "Service axis" function opens.
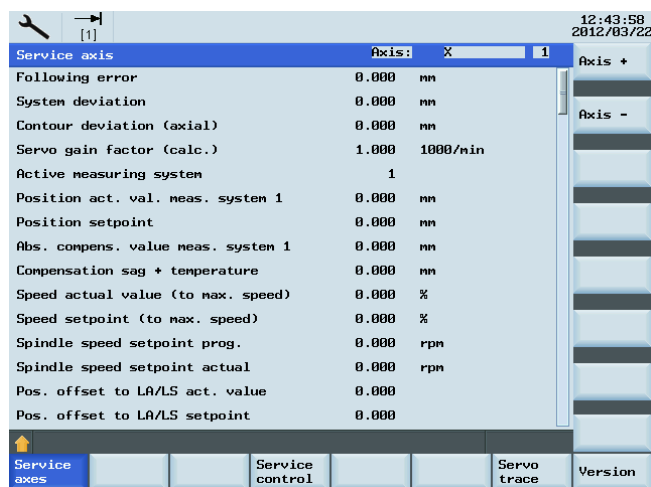
You can select your desired service function. By default, this window displays information on the axis drive. The "Axis +" and "Axis -" softkeys are additionally displayed. These can be used to display the values for the next or previous axis.

3. To view the action log for service events, press the "Service control" softkey. See the Section "Action log (Page 17)" for more information.

4. By pressing the "Servo trace" softkey, you can compare the defined machine data with the actual data and adjust the parameters if necessary. The oscilloscope function is available in the "Servo trace" window to optimize the drives. See the Section "Servo trace (Page 22)" for more information.

5. Press the "Version" softkey to display the version numbers and the date of creation of the individual CNC components.

The following functions can be selected from this window (see section "Version/HMI details (Page 27)"):

- "HMI details"
- "License key"
- "Options"
- "Save as..."

2.4.1 Action log

Overview

The function "Action log" is provided for service events. The contents of the action log file can only be accessed through a system password on the HMI.

Viewing the action log

To view the action log, follow these steps:

1. On the "Service control" main screen, press the "Action log" softkey.
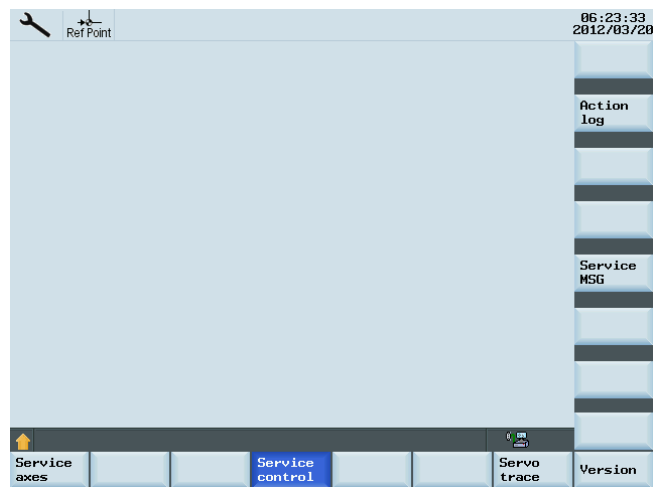
2. The "Action log" window opens. Three softkeys are available: "Setting", "Search" and "Save as...". Choose your desired option.

    To search for a specific event, press the "Search" softkey.

    To save the selected action log on a CF card or on a USB memory stick, press the "Save as..." softkey.
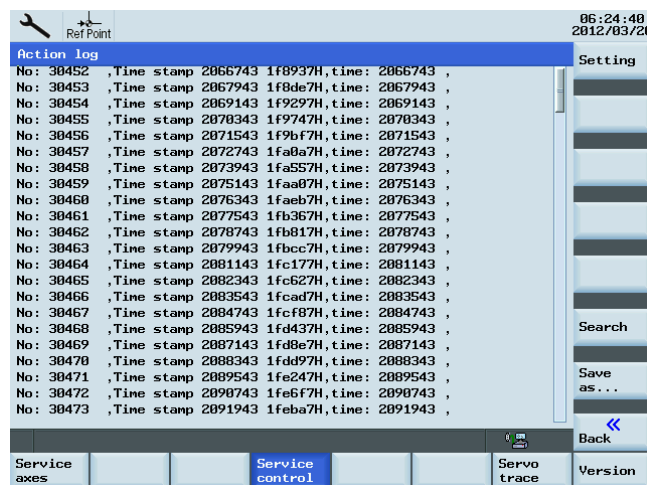
3. Press the "Setting" softkey to set the filter conditions for the data to be displayed.
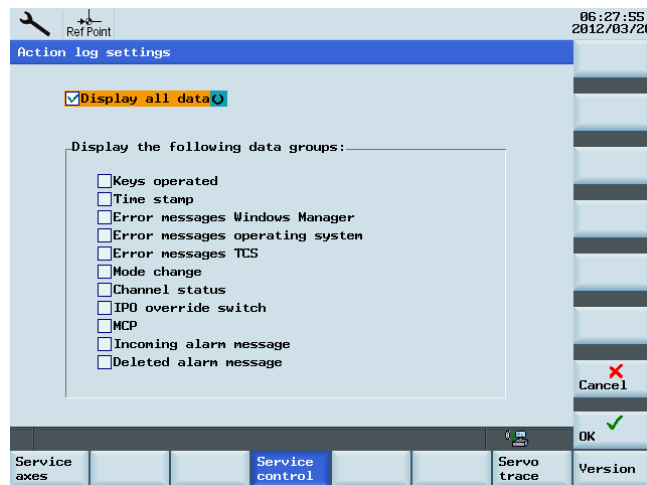
4. Press the **<TAB>** key to switch between displaying all data and displaying a data group. Press the **<SELECT>** key to select or deselect the displayed data.

2.4.2 Service MSG

Overview

The "Service MSG" function provides the possibility of sending message texts / messages to a PG/PC via the RS232 interface and/or to a file to be saved internally on the control system.

Message texts / messages include:

- Programmed messages from part program
- Alarms

Settings for the message sending

1. Press the "Service MSG" softkey on the "Service control" main screen.
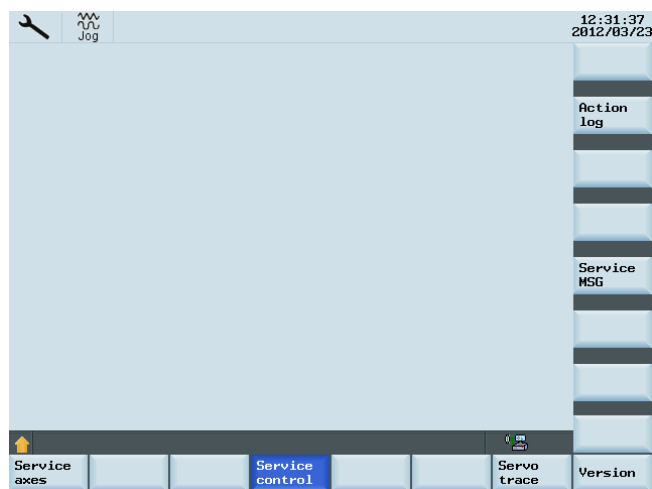
2. The "Service MSG" window opens.
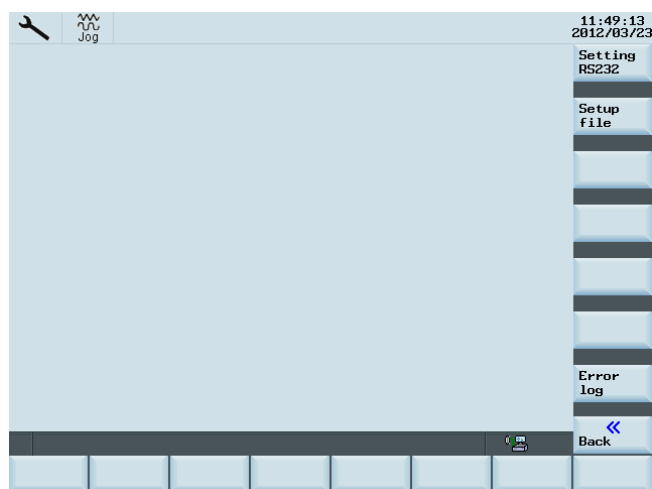
3.   • Press "Setting RS232" to open the "RS232 settings" window.

Press ▲ and ▼ to move selection up and down. Press <SELECT> to select or deselect the settings for sending the messages via the RS232 interface. Press "OK" to save the settings and return to the "Service MSG" window.
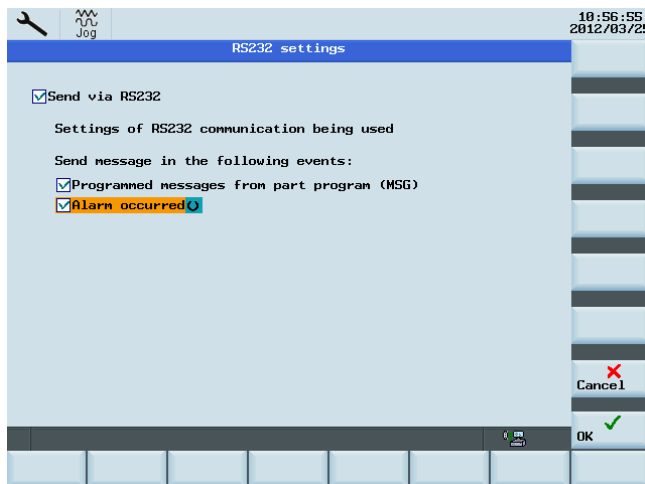
• Press "Setup file" to open the "File settings" window.

Press ▲ and ▼ to move selection up and down. Press <SELECT> to select or deselect the settings for the file to which you desire to save the messages, and input the file name, for example, TEST.TXT. Press "OK" to save the settings and return to the "Service MSG" window.
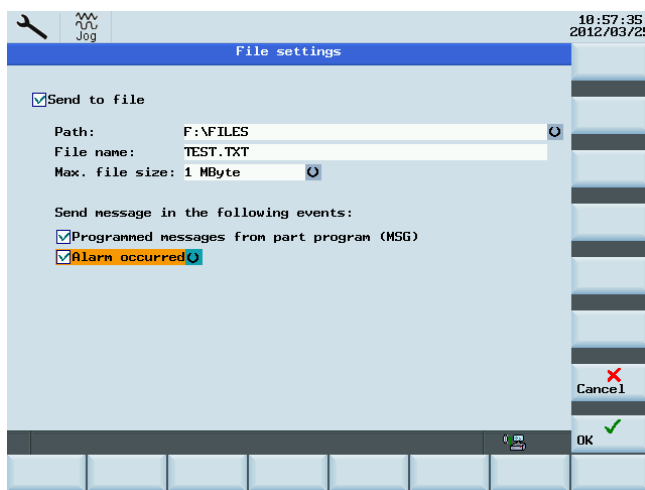
You can then find the message file under the path selected:
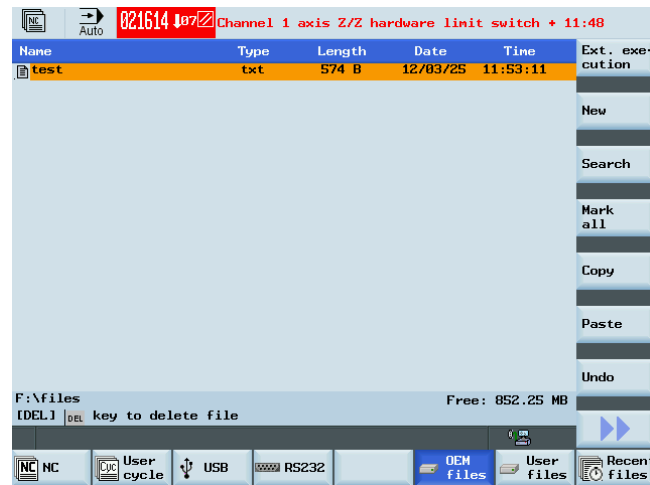
Note

To transfer messages via the RS232 interface, the communication settings from the "SYSTEM" operating area > "Sys. data" > "RS232" > "Settings" are used. Refer to Section "External data backup through the RS232 interface (Page 310)" for detailed settings for RS232 communication.

When using the MSG service via RS232, the RS232 interface must not be active for another application. This means, e.g. the RS232 interface must not be active from the "SYSTEM" operating area > "PLC" > "STEP7 connect".

Viewing the error log

All messages with the associated error information, where an error occurred when processing them, are saved in the error log. To view the error log, press "Error log" in the "Service MSG" window and the "Error log MSG command" window opens. The error log can be deleted using the "Delete log" softkey.
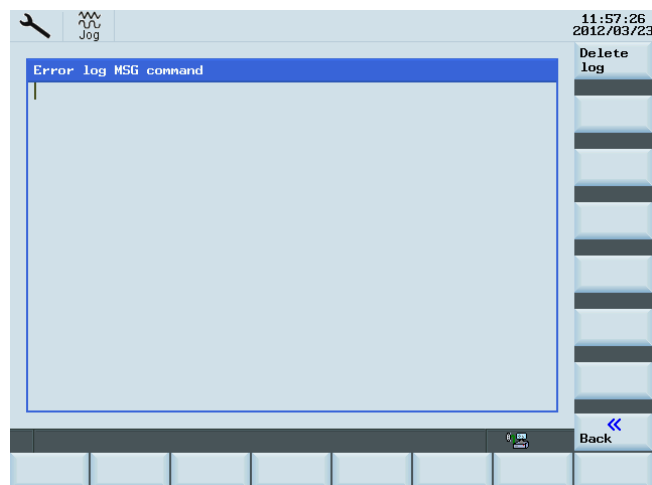
2.4.3   Servo trace

Overview

An oscilloscope function is provided for the purpose of optimizing the drives. This enables the following graphical representations:

- velocity setpoint

- contour violation

- following error

- actual position value

- position setpoint

- exact stop coarse / fine

Viewing the servo trace

To view the servo trace, follow these steps:

1. Press "Servo trace" on the service display start screen.
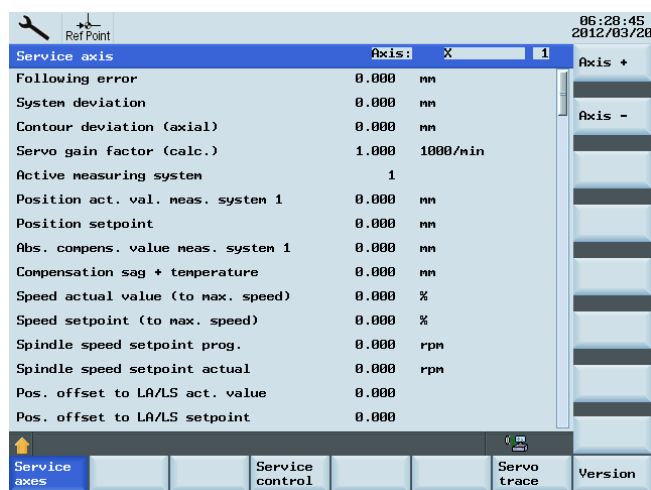
2. The servo trace main screen opens. To analyze the result, you can perform the following operations:

- Changing and scaling the abscissa and ordinate values

- Measuring a value using the horizontal or vertical marker

- Measuring the abscissa and ordinate values as a difference between two markers

- Storing the result as a file in the part program directory
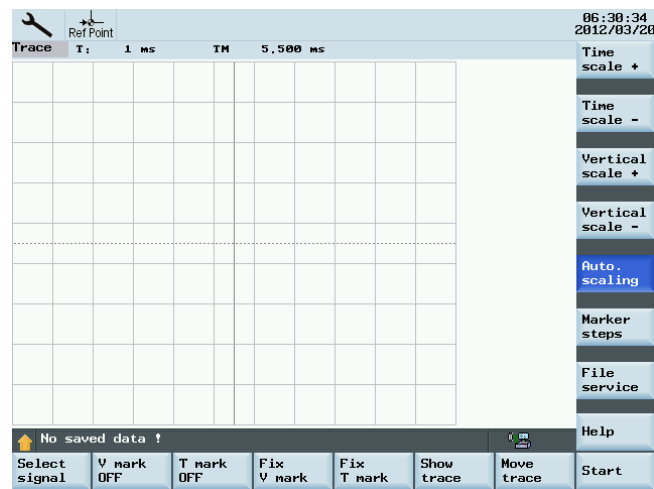
The header of the diagram contains the current scaling of the abscissa and the difference value of the markers.
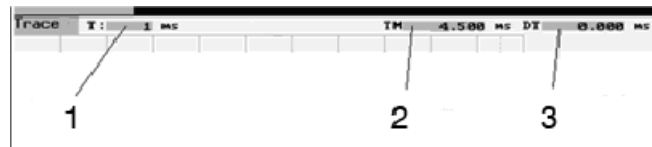
1   Time Base
2   Marker position time
3   Difference in time between marker 1 and current marker position

The following table shows the description of softkeys on the main screen in detail:

Softkey name	Function
Select signal	Parameterizes the measuring channel
V/T mark ON/OFF	Hides/shows the vertical or time gridlines
Fix V/T mark	Determines the differences in the horizontal or vertical directions. After selected, the softkey labels change to "Released V mark" or "Released T mark".
Show trace	Hides / displays the selected diagrams in various traces.
Move trace	Moves the trace.
Start/stop	Starts or stops a servo trace.
Time scale+/-	Zooms in / zooms out the time basis.
Vertical scale+/-	Increases / reduces the resolution (amplitude).
Auto. scaling	Shows the vertical or time grindlines automatically.
Marker steps	Defines the step sizes of the markers.
File service	Saves or loads the trace data
Help	Provides the help info.

Selecting signal

To parameterize the measuring channel, follow these steps:

1. Press "Select signal" to open the "Signal selection" main screen.
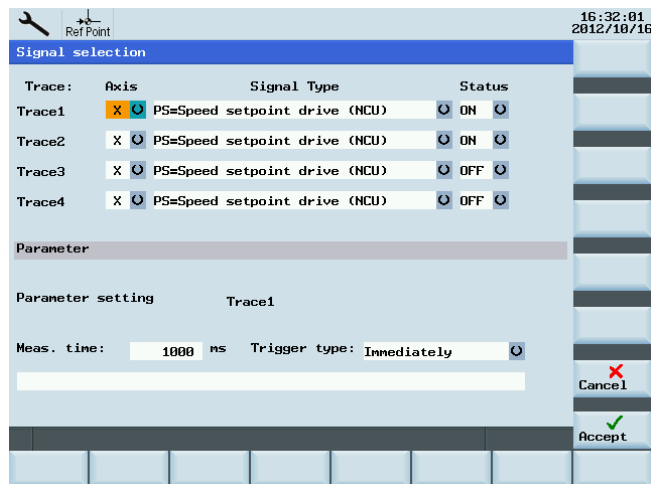
2. In the upper screen half, it shows the axis, signal type and status of the traces. Press **<SELECT>** to choose the axis, signal type and enable tracing in the respective traces.

3. In the lower screen half, set the parameters for the measuring time and for the trigger type for channel 1. The setting also applies to other traces.

4. Enter the measuring period in the "Meas. time" field (6,133 ms max).

5. Position the cursor to the "Trigger type" filed and press **<SELECT>** to select one of the following trigger types:

- Immediately
- Positive edge
- Negative edge

The following signal types are provided:

- Following error
- System deviation
- Contour deviation
- Actual position value
- Position setpoint
- Actual speed value encoder
- Speed setpoint drive
- Compensation value
- Parameter block
- Input of position setpoint controller
- Input of speed setpoint controller
- Input of acceleration actual value controller
- Speed feedforward value plus QEC
- Exact stop fine
- Exact stop rough

Defining the marker steps

To define the marker steps, follow these steps:

1. Press "Marker steps" on the servo trace main screen.

2. The marker increment window opens. Select the desired increment value from the vertical softkeys "V=0.1/0.5/1.0" or "T=0.1/0.5/1.0". For other step sizes, press "Other".
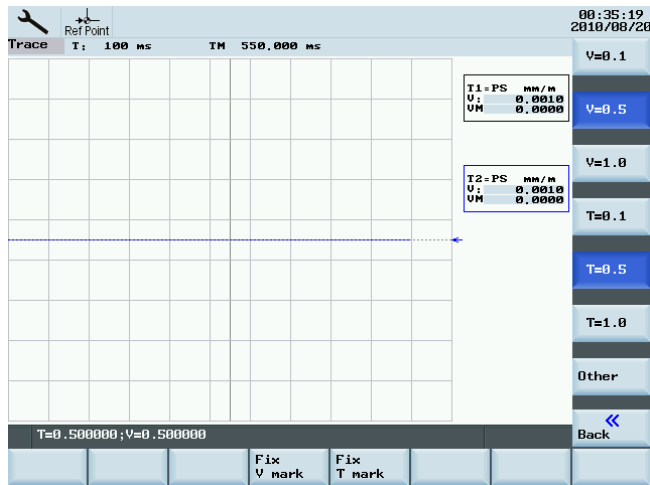
3. If the trace exceeds the current screen, press <SHIFT> + cursor movement. When a marker reaches the margin of the diagram, the grid automatically appears in the horizontal or vertical direction.
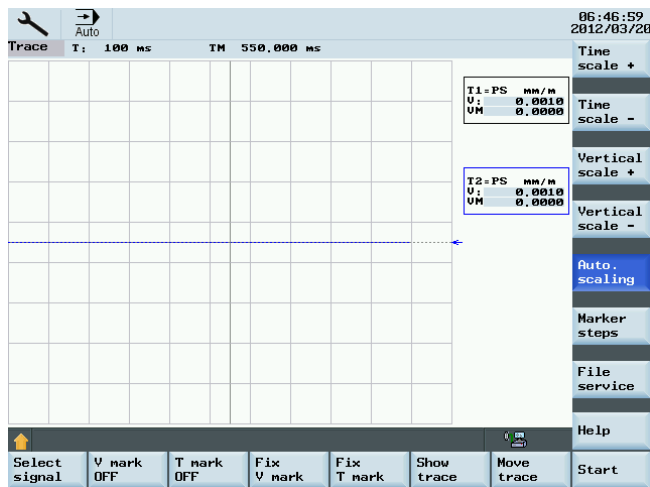
Saving trace data

To save the trace data, follow these steps:

1. Press "File service".
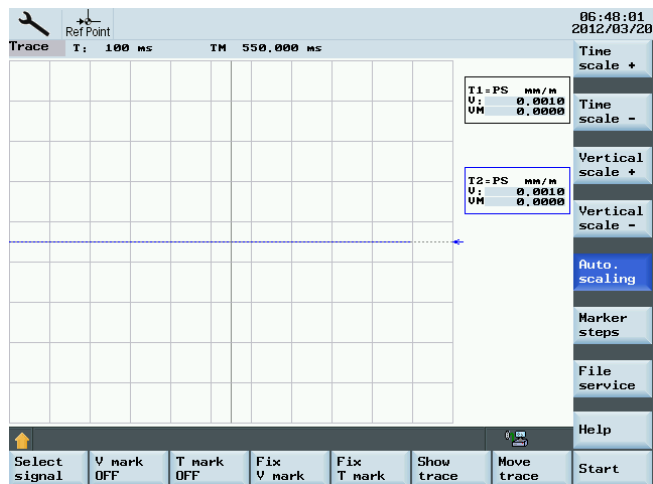
2. Type the desired file name without extension in the "File name" field.

3. Use the "Save" softkey to save the data with the specified name in the part program directory, and the file can be exported, and the data can be processed in MicroSoft Excel.

4. Press "Load" to load the specified file and graphically displays the data.
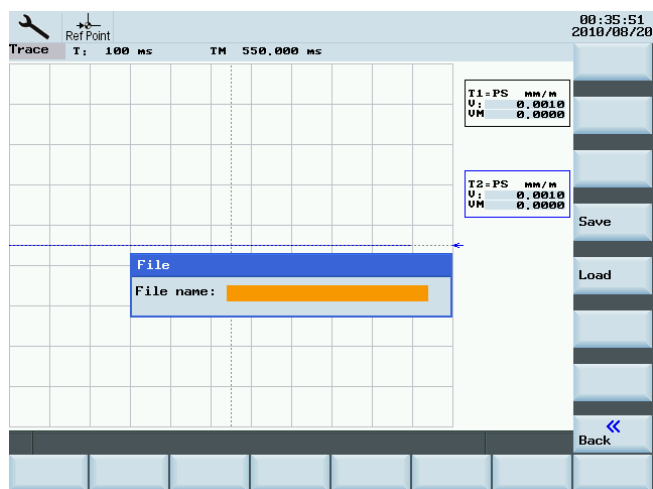
2.4.4 Version/HMI details

Overview

The "Version" window displays the version numbers and the date of creation of the individual CNC components. Four softkeys "HMI details", "License key", "Options", and "Save as" are available, which enable the following operations:

- Display the operator programs with the version numbers.
- Enter the license key.
- Save the version data.
- Activate Additional Axis and Manual Machine Plus (valid only for the turning variant of the control system)
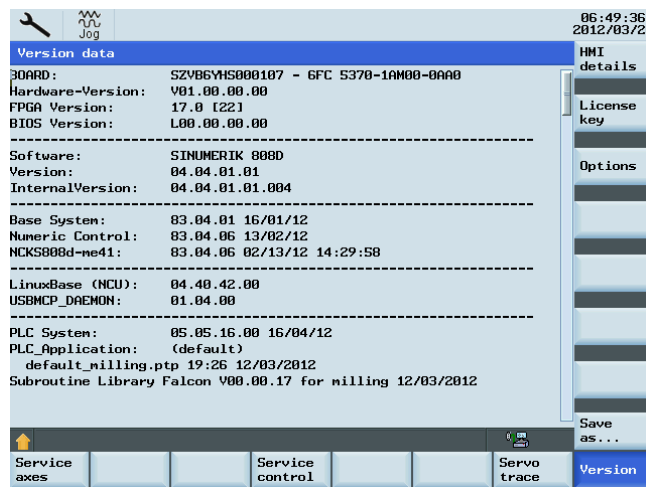
HMI details

The "HMI details" menu is intended for servicing and can only be accessed via the user password level. All programs provided by the operator unit are displayed with their version numbers. By reloading software components, the version numbers can be different from each other.

Press "HMI details" on the Version main screen, and then the "DLL details" window opens.
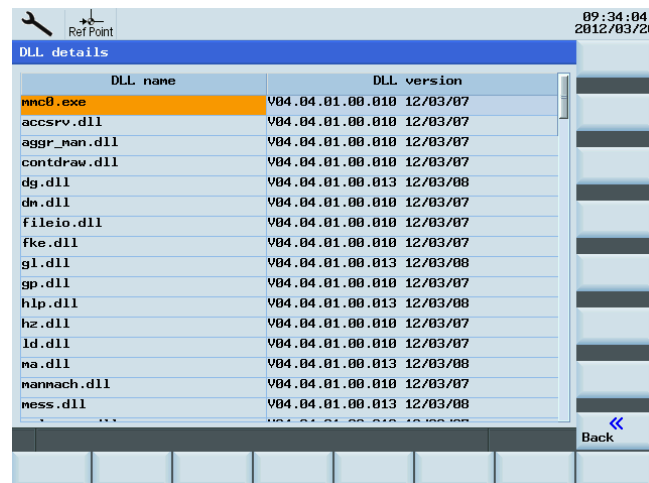
License key

To enter the license key, follow these steps:

1. Press "License key".
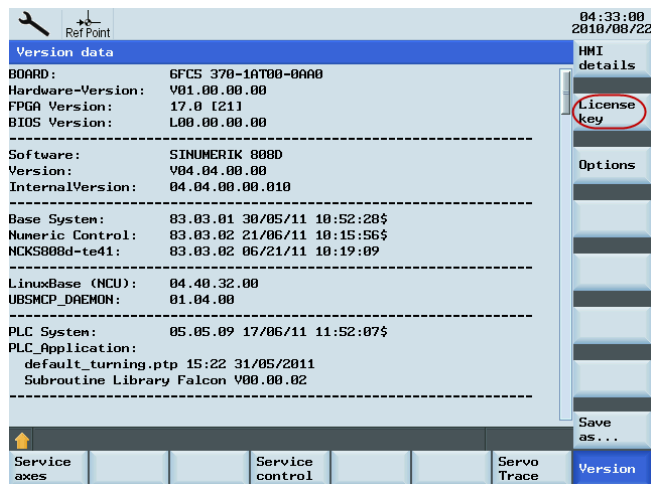
2. Enter the license key and then press "OK".
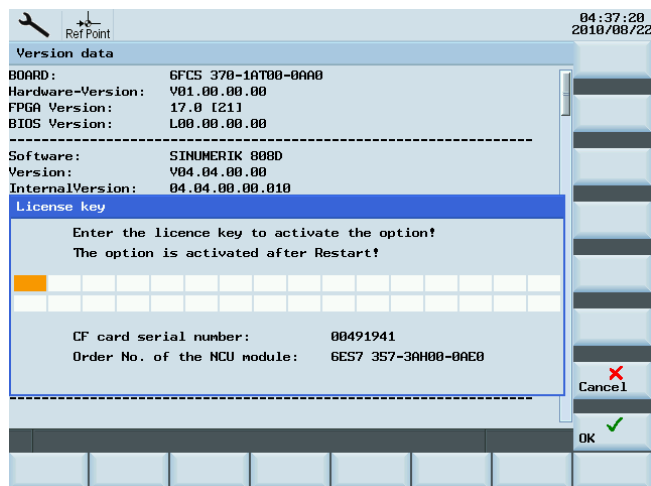
Options

> **Note**
>
> The software options "Additional Axis" and "Manual Machine Plus" are only used for the turning variant of the control system. They are not available on the milling variant of the control system. To use these options, you must first activate them by entering the valid license key.

Proceed as follows to configure the Additional Axis and Manual Machine Plus functions:

1. Press "Options" on the Version main screen to open the "Options" window.
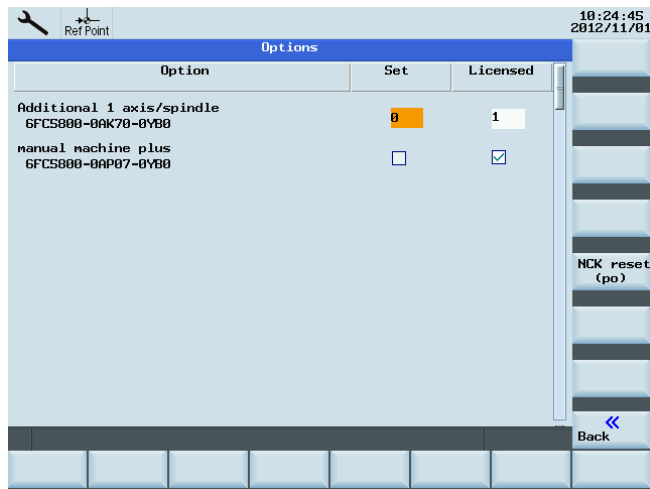
2. If you have activated the options by entering the licence key, the "Licensed" column shows value "1" for the Additional Axis and " ☑ " for Manual Machine Plus.

  – For the Additional Axis, input "0" / "1" in the "Set" column to disable / enable the function

  – For the Manual Machine Plus, press <SELECT> in the "Set" column to select or deselect the function

3. Press the softkey "NCK reset (po)" to restart the NCK.

Reference

SINUMERIK 808D Commissioning Manual

2.5   PLC diagnostics

Functionality

A PLC user program consists to a large degree of logical operations to realize safety functions and to support process sequences. These logical operations include the linking of various contacts and relays. As a rule, the failure of a single contact or relay results in a failure of the whole system/installation.

To locate causes of faults/failures or of a program error, various diagnostic functions are offered in the "SYSTEM" operating area.

Operating sequence

1. Press [SHIFT] + [SYSTEM ALARM] to enter the "SYSTEM" operating area.

2. Press "PLC" > "PLC program" to open the project stored in the permanent memory.
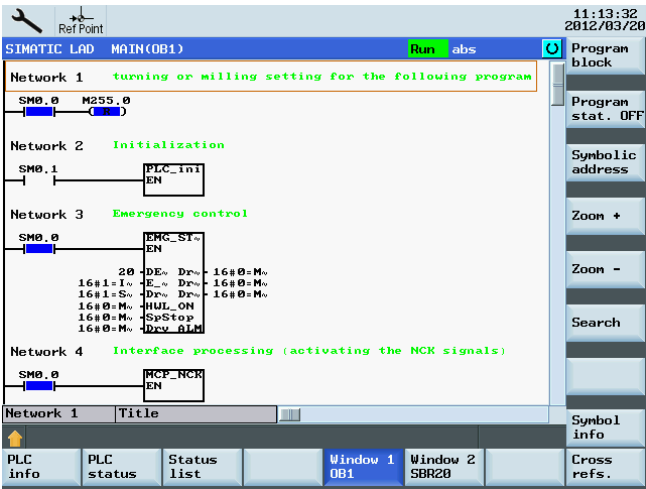
2.5.1 Screen layout
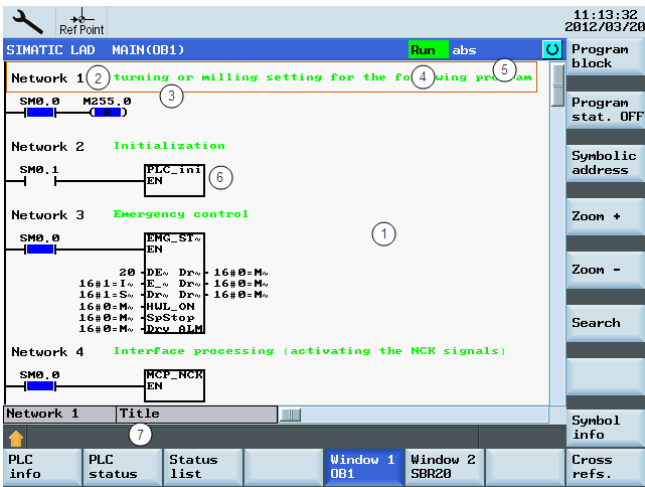
Figure 2-2   Screen layout

Table 2-2   Legend description

Legend	Display	Meaning
①		Application area
②		Supported PLC program language
③		Name of the active program block
		Representation: Symbolic name (absolute name)
④		Program status
	RUN	Program is running
	STOP	Program stopped
	Status of the application area	
	Sym	Symbolic representation
	abs	Absolute representation

Table 2-2   Legend description

Legend	Display	Meaning
⑤		**Display of the active keys**
⑥	**Focus** Performs the tasks of the cursor	
⑦	**Tip line** contains notes for searching	

Softkeys

The following table describes the softkeys on the default screen as shown above.

Softkey	Function
PLC info	Displays PLC properties
PLC status	Monitors and changes the values of the operands during program execution.
Status list	Displays and modifies PLC signals
Window 1	Displays any logical and graphical information of a program block
Window 2	Displays any logical and graphical information of a program block in another window
Cross refs.	Displays the list of cross-references
Program block	Displays the logic and graphic information of the selected program block.
Program stat.OFF	Activates or deactivates the program status display. The states of all operands are displayed in the "Program status" ladder diagram (top right in the window).
Symbolic address	Switches between the absolute and symbolic representation of the operands. The softkey labelling changes accordingly.
Zoom +	Zooms in the application area
Zoom -	Zooms out the application area
Search	Searches for operands
Symbol info	Displays all symbolic identifiers used in the selected network

2.5.2   Operating options

In addition to the softkeys and the navigation keys, this area provides still further key combinations.

Hot keys

The cursor keys move the focus over the PLC user program. When reaching the window borders, it is scrolled automatically.

Table 2-3   Hot keys

Keystroke combination	Action
NEXT WINDOW or CTRL ◀	To the first line of the row
END or CTRL ▶	To the last line of the row
PAGE UP	Up a screen
PAGE DOWN	Down a screen
◀	One field to the left
▶	One field to the right
▲	Up a field

Table 2-3   Hot keys

Keystroke combination	Action
▼	Down a field
CTRL NEXT WINDOW or CTRL ▲	To the first field of the subroutine
CTRL END or CTRL ▼	To the last field of the subroutine
CTRL PAGE UP	Opens the previous program block in the same window
CTRL and PAGE DOWN	Opens the next program block in the same window

Table 2-3   Hot keys

Keystroke combination	Action
SELECT	The function of the Select key depends on the position of the input focus. • Table line: Displays the complete text line • Network title: Displays the network comment • Command: Displays the complete operands
INPUT	If the input focus is positioned on a command, all operands including the comments are displayed.

Displaying the PLC info

On the PLC program start screen, press "PLC info" to view the following PLC properties:

• Operating status

• Project name and change info

• PLC version and system info

• Cycle time

• Processing time of the PLC user program
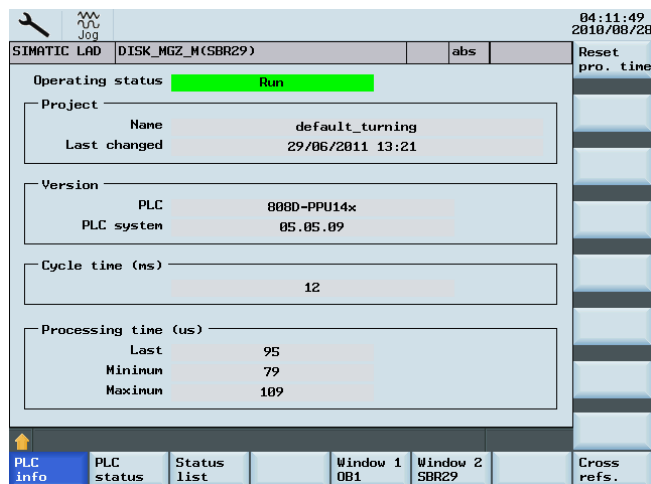
By pressing the "Reset pro. time" softkey, the processing time data is reset.

Searching for operands

In big programs, you can use the search function to quickly reach the desired positions.

To search for operands, follow these steps:

1. Use the "Absolute address" or "Symbolic address" to switch between the absolute and symbolic representation of the operands.
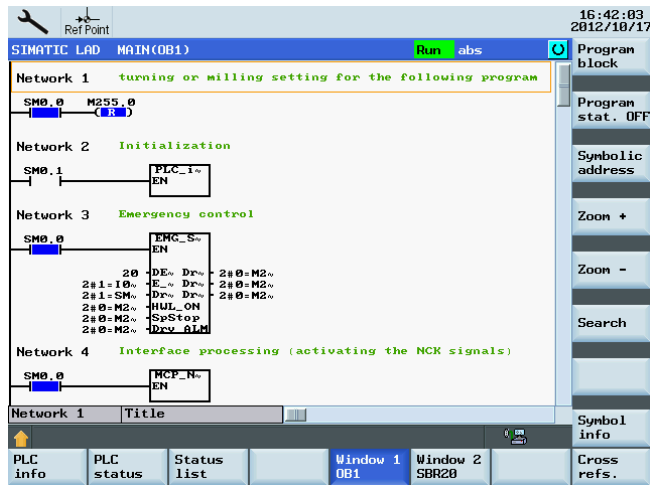
2. Press "Search" to open the following window.
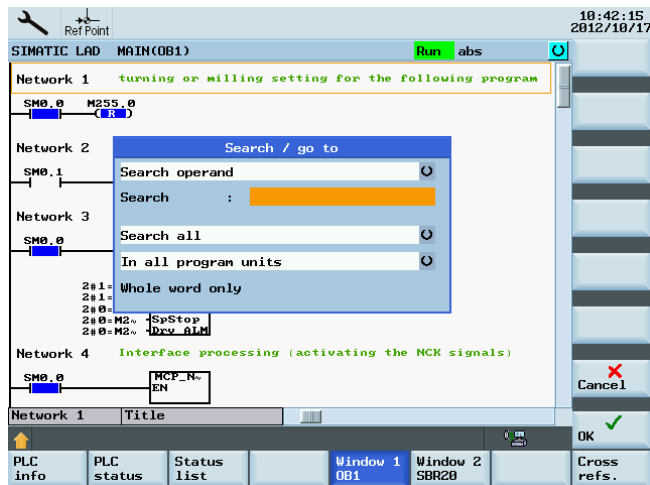
3. Press the <SELECT> key on the PPU to select the desired search criterion from the options: "Search operand", "Search subroutine command" or "Go to".

4. Enter the search term in the "Search" field.

5. Select the search range (e.g. "Search all").

6. Select "In this program unit" or "In all program units" if you are in "Window 1" or " Window 2".

7. Press "OK" to start the search.

8. Press "Cancel" to exit.

2.5.3 Displaying information on the program blocks

Functionality

Using the "Window 1 ..." and "Window 2 ..." softkeys, you can display any logical and graphical information of a program block. The program block is one of the components of the PLC user program.
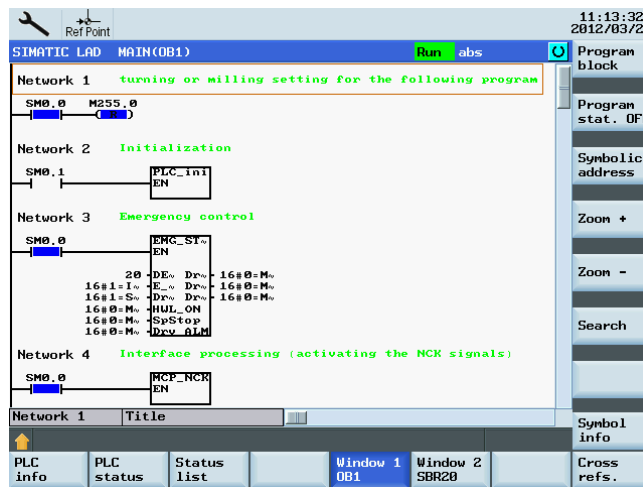
Logic information

The logics in the ladder diagram (LAD) display the following:

- Networks with program parts and current paths
- Electrical current flow through a number of logical operations

Further information

- Properties

Symbolic name, author and comments

- Local variables

Name of the variable, variable type, data type and comment

Procedure

1. Press "Program block". The "Program block" window opens.
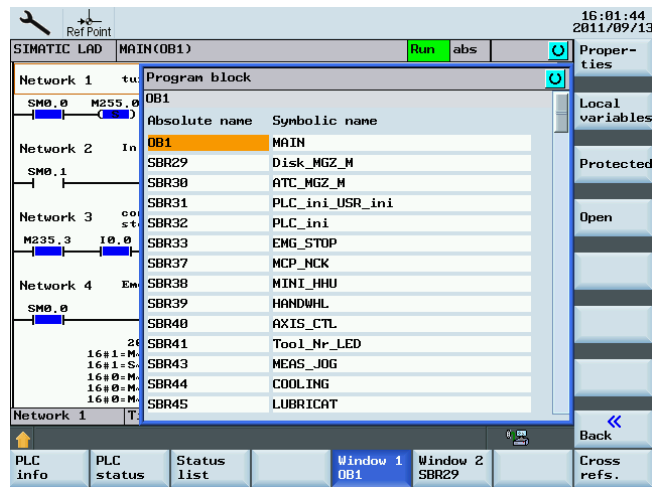
2. If a program block is protected by a password, press "Protected" to enable the display of the ladder diagram. A password is required for this. The password can be allocated during creation of a program block in Programming Tool PLC808.

3. Select the desired program block and press "Open". The name of the program block will be displayed on the softkey (e.g. "Window 1 OB1").

4. The selected program block is opened. You can press "Absolute/Symbolic address" to switch the display information. The screenshot shows the symbolic address when the program status is on.
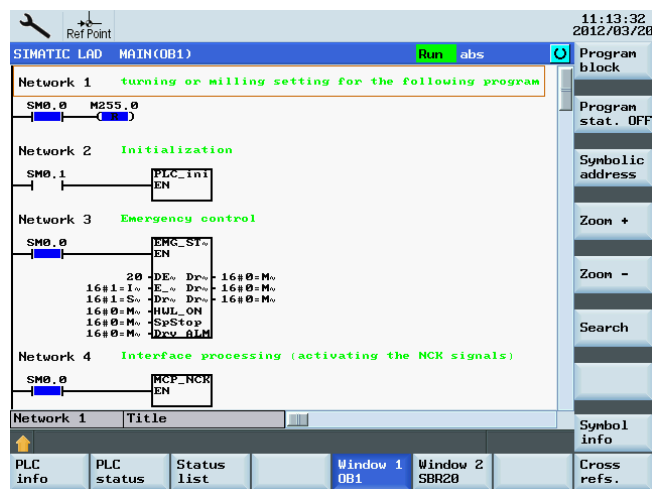
5. In the "Program block" window, press "Properties" if you want to display additional information; press "Local variables" if you want to display the table of local variables of the selected program block.

2.5.4   Displaying cross-references

Functionality

Use the "Cross refs." softkey to display the list of cross references. All operands used in the PLC project are displayed.

This list indicates in which networks an input, output, flag etc. is used.

To display the cross-references, follow these steps:

1. Press "PLC program".

2. Press "Cross refs.". The list of cross references appears and the operands are displayed according to the absolute address by default.

3. Press "Absolute/Symbolic address" to switch between the absolute and symbolic representation of the components.

4. You can open the appropriate program segment directly in the 1/2 window using the "Open in window 1" or "Open in window 2" function. The screenshot uses window 1 as an example.
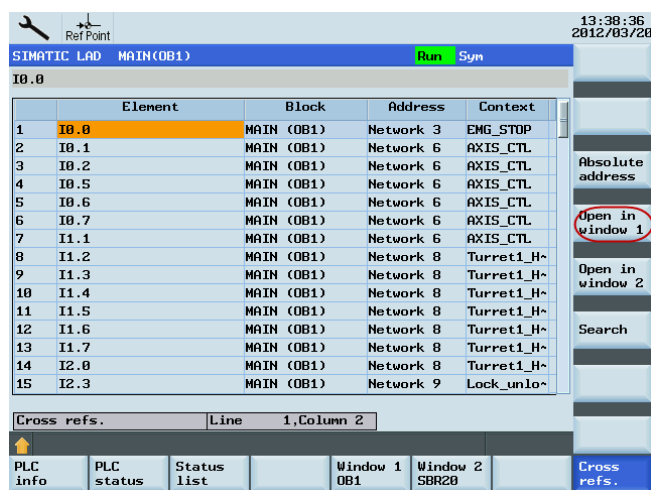
5. After the operand has been selected from the cross-reference list and the "Open in window 1" softkey has been pressed, the corresponding program section is displayed in window 1.
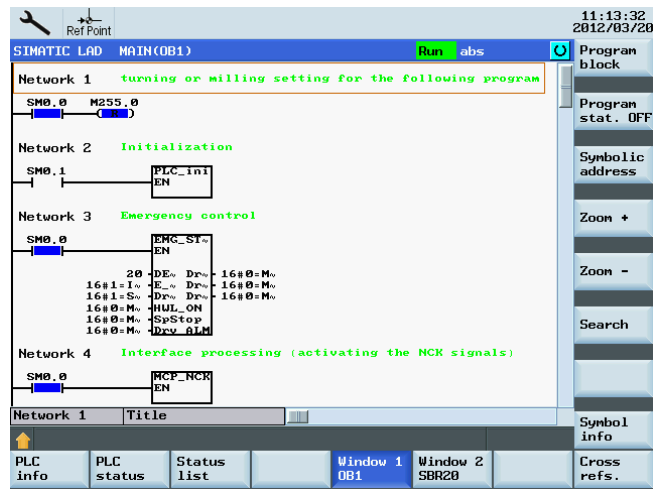
6. If you want to search operands in the cross-reference list, press "Search" to open the "Search / go to" window.
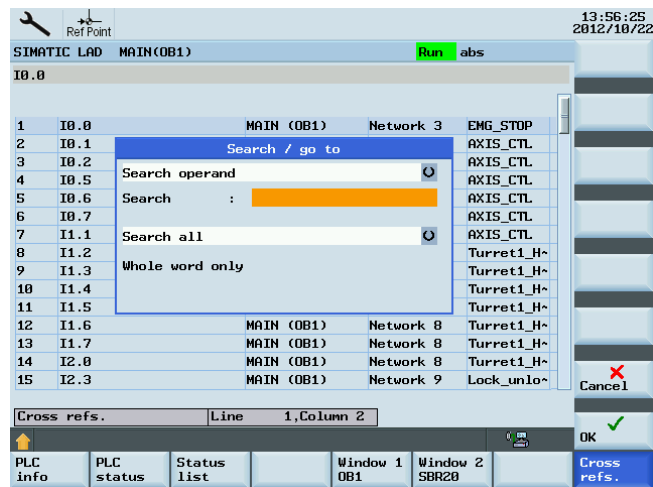
7. Press <SELECT> to select "Search operand" or "Go to", and then enter the desired element or line.

8. Press "OK" to start the search; press "Cancel" to exit.

2.6 Defining the maintenance planner

Overview

This part introduces how to define the OEM maintenance planner. By pressing the "Serv. planr." softkey, you can specify the maintenance timer and define your maintenance info.

Main screen of "Service planner"

The main screen of "Service planner" displays the position, task description, interval, first warning time, number of warnings, etc.
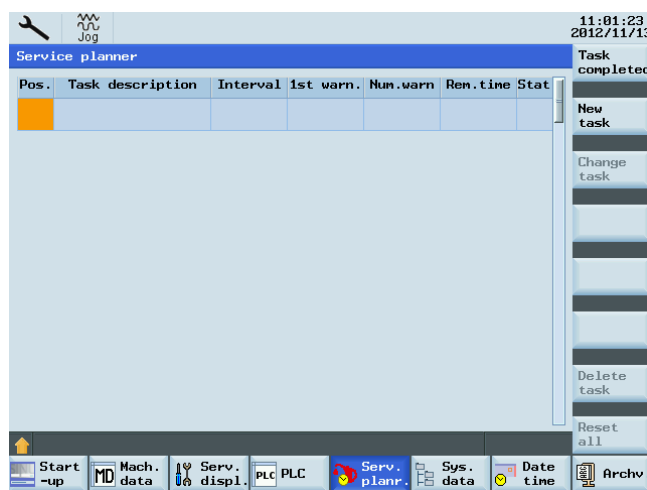
The following table describes the softkeys on the main screen:

Softkey	Function
Task completed	Completes
New task	Creates a new task
Change task	Changes the selected task
Delete task	Deletes the selected task
Reset all	Resets the remaining time for all tasks

Creating a new maintenance task

To create a new maintenance task, follow these steps:

1. Press the "New task" softkey, and the "Create new task ..." window opens.

2. Enter values in respective fields and then press "OK".

2.7 Managing the system data

Overview

By pressing the "Sys. data" softkey in the "SYSTEM" operating area, you can enter the "808D data" window.

Two folders, "HMI data" and "NCK/PLC data", and one "File for license key" are available in this window. You can import the HMI data in separate files or subfolders for customized purposes.

Importing HMI data

Select "HMI data" in the "808D data" window and press the <INPUT> key to enter. There are 11 folders available on the "HMI data" screen as follows:

```
⭤ ..
🗁 Customized bitmaps
🗁 User cycle files
🗁 EasyXLanguage scripts
🗁 OEM online help (*.txt;*.png;*.bmp)
🗁 Extended user text file (almc....txt)
🗁 OEM MD description file (md_descr....txt)
🗁 OEM manual (oemmanual.pdf)
🗁 PLC alarm texts (alcu....txt)
🗁 OEM slideshow (*.bmp;*.png)
🗁 OEM R variable name file (rparam_name....txt)
🗁 Service planner task name file (svc_tasks....txt)
```
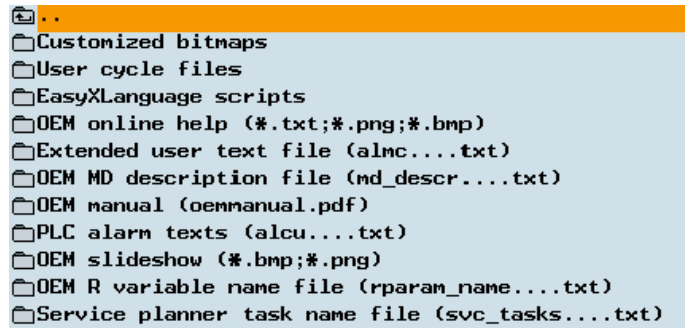
Customized bitmaps

You can use you own start-up screen and machine logo on the control system by importing your own files in the BMP format into this folder.

For more information, refer to the SINUMERIK 808D Commissioning Manual.

User cycle files

This folder is used to store the bitmap file, alarm file, softkey index file and parameter file required for creating a user cycle.

For more information, refer to the SINUMERIK 808D Function Manual.

EasyXLanguage scripts

The SINUMERIK 808D offers an XML-based script language for generating user dialogs. This script language makes it possible to display machine-specific menus and dialog forms in the "CUSTOM" operating area on the HMI.

For more information, refer to the SINUMERIK 808D Function Manual.

OEM online help

Besides using the Siemens online help on the control system, you can also create your own online help files and import them into the control system.

For more information, refer to the SINUMERIK 808D Function Manual.

Extended user text file

The extended user text file is required for the display of respective screen texts, cycle messages and softkey texts for your customized cycle.

For more information, refer to the SINUMERIK 808D Function Manual.

OEM MD description file

By importing an OEM MD description file into this folder, you can apply your own machine data descriptions. When creating the file, the following file naming rule must be observed:

md_descr_xxx.txt

Here "xxx" refers to the language denotation, for example, "eng". You can create the OEM MD description file in English, Chinese, Portuguese, or Russian but only under the corresponding system language environment.

Copy the created files into this folder using a USB stick, and then you can see your customized MD descriptions from the "SYSTEM" operating area > "Mach. data".

OEM manual

You can copy your own manuals in the PDF format into this folder. The manuals can then be called via the "OEM Manual" softkey in the help system.

To import an OEM manual using a USB stick, proceed as follows:

1. Since the OEM manual supports four languages, you first need to create a three-level nested folder of your desired language on the USB stick according to the following structure.

   First level:

⤴ ..			
📁chs	DIR	11/10/11	00:19:18
📁eng	DIR	11/10/11	00:19:18
📁ptb	DIR	11/10/11	00:19:18
📁rus	DIR	11/10/11	00:19:18

   Second level:

⤴ ..			
📁milling	DIR	11/10/11	00:19:18
📁turning	DIR	11/10/11	00:19:18

   Third level:

⤴ ..			
📁manual	DIR	11/10/11	00:20:33

2. Create your own manual in the PDF format and copy it to the third-level folder in the USB stick.

3. Copy the first-level folder(s) from the USB stick to the "OEM manual (oemmanual.pdf)" folder on the control system.

   Then you can view your own manual by pressing the <HELP> key and then pressing the "OEM Manual" softkey.

PLC alarm texts

By importing a PLC alarm text file into this folder, you can apply your own PLC alarm texts. When creating the file, the following file naming rule must be observed:

alcu_xxx.txt

Here "xxx" refers to the language denotation, for example, "eng". You can create the PLC alarm text file in English, Chinese, Portuguese, or Russian but only under the corresponding system language environment.

Copy the created files into this folder using a USB stick, and then you can see your customized PLC alarm texts from the "SYSTEM" operating area > "PLC" > "Edit PLC alarm txt".

OEM slideshow

The SINUMERIK 808D has a function of playing a slide show. You can create a slide show to display your product information.

For more information, refer to the SINUMERIK 808D Commissioning Manual.

OEM R variable name file

By importing a R variable name file into this folder, you can apply your own R variable names. When creating the file, the following file naming rule must be observed:

rparam_name_xxx.txt

Here "xxx" refers to the language denotation, for example, "eng". You can create the OEM R variable name file in English, Chinese, Portuguese, or Russian but only under the corresponding system language environment.

Copy the created files into this folder using a USB stick, and then you can see your customized R variable names from the "OFFSET" operating area > "R var." > "Show R name".

Service planner task name file

The SINUMERIK 808D has the service planning function. With this function, you can specify the service timer and define your own service information.

For more information, refer to the SINUMERIK 808D Commissioning Manual.

2.8 Archive setting up

Functionality

You can create or restore a start-up or series start-up archive in the "Start-up archive" window.
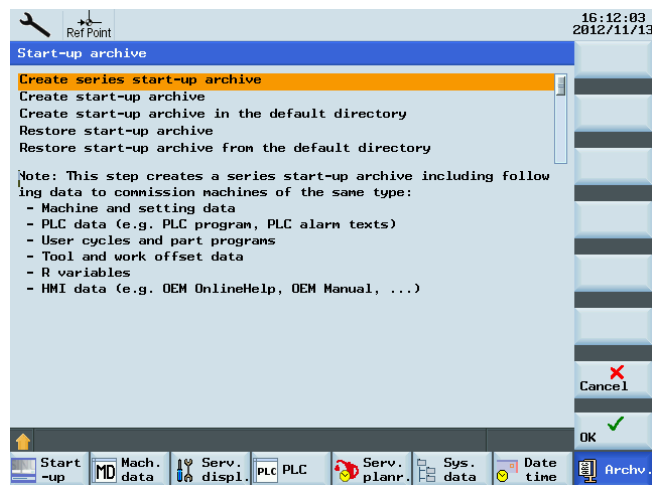
Procedure

1. In the "SYSTEM" operating area, press the horizontal "Archv." softkey to open the "Start-up archive" window.

2. Use the cursor keys to create or restore the start-up archive.

3. Press "OK" to confirm; press "Cancel" to cancel.

2.9 Alarm display

Operating sequence

The alarm window is opened. You can sort the NC alarms using the softkeys. PLC alarms are **not** sorted.
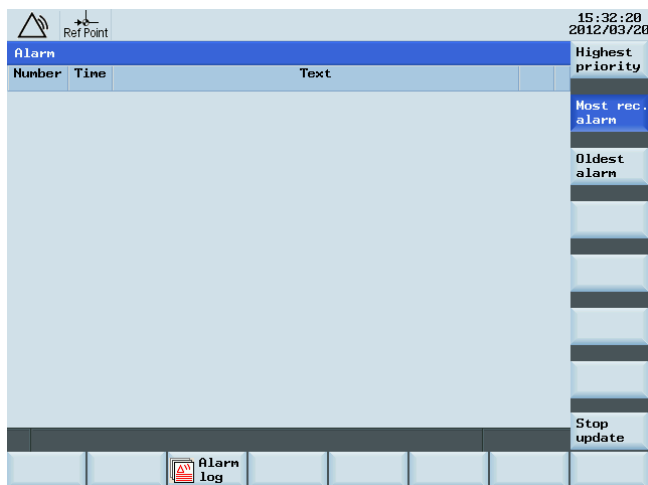
Softkeys

Softkey	Function
Highest priority	Displays all alarms sorted by their priorities. The highest priority alarm is at the beginning of the list
Most rec.alarm	Displays the alarms sorted by the time of their occurrence. The most recent alarm stands at the beginning of the list.
Oldest alarm	Displays the alarms sorted by the time of their occurrence. The oldest alarm stands at the beginning of the list.
Stop update	Stops updating of pending alarms
Alarm log	Views and manages the alarm log.

Viewing and managing alarm logs

To view and manage the alarm log, follow these steps:

1. Press "Alarm log". Scroll up and down to view your desired alarm log.

2. Press "Save as..." to save the file to other storage media.

3. Press "Delete log" to delete all alarm logs.

SINUMERIK 808D alarms

<div style="text-align: right; font-size: 3em;">3</div>

3.1 System error alarms

The following alarms are system errors:

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

These system error alarms are not described in detail. If such a system error occurs, contact the hotline and indicate the following details:

- Alarm number

- Alarm text

- Internal system error number (contained in the alarm text)

3.2 NCK alarms

2000   **PLC sign-of-life monitoring**

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

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

Remedy: Please inform the authorized personnel/service department. Check monitoring time frame in MD10100 $MN_PLC_CYCLIC_TIMEOUT (reference value: 100ms).
Establish the cause of the error in the PLC and eliminate it (analysis of the ISTACK. If monitoring has responded with a loop in the user program rather than with a PLC Stop, there is no ISTACK entry).
This alarm is also caused by PLC stop.
(PLC stop with programming tool,
PLC stop by commissioning switch,
PLC stop by alarm)
If none of these cases applies, place a support request with the error text under: http://www.siemens.com/automation/ support-request

Program Continuation: Switch control OFF - ON.

2001	PLC has not started up
Definitions:	The PLC must give at least 1 sign of life within a period of time defined in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT (default: 1 sec.). The PLC must give at least 1 sign of life within the defined period of time after Power ON.
Reaction:	NC not ready. Local alarm reaction. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	- Please inform the authorized personnel/service department. The monitoring time in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT must be checked and adapted to the first OB1 cycle. - Determine the cause of error in the PLC (loop or stop in the user program) and eliminate it. Place a support request with the error text under: http://www.siemens.com/automation/support-request
Program Continuation:	Switch control OFF - ON.

2100	NCK battery warning threshold reached
Definitions:	The undervoltage monitor of the NCK battery has reached the prewarning threshold.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. The battery in the NC module must be replaced within the next 6 weeks (see Manual) in order to avoid loss of the buffered memory.
Program Continuation:	Clear alarm with the Delete key or NC START.

2101	NCK battery alarm
Definitions:	The undervoltage monitoring of the NCK battery has responded during cyclic operation.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Exchange the battery in the NC module without interrupting the power supply (see Manual) in order to avoid loss of the buffered memory.
Program Continuation:	Clear alarm with the Delete key or NC START.

2102	NCK battery alarm
Definitions:	The undervoltage monitoring of the NCK battery was detected during system power-up.
Reaction:	NC not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Exchange the battery in the NC module (see Manual). The system then has to be reinitialized because it has to be assumed that the buffered memory has been lost as a result of an inadequate supply voltage during the last power-off phase (see Commissioning Manual).
Program Continuation:	Switch control OFF - ON.

2110	NCK temperature alarm
Definitions:	The temperature sensor has reached the response threshold.
Reaction:	Alarm display.
Remedy:	In order to reset the sensor, the temperature must be reduced by 7 degrees C.

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

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

Definitions:	A failure has occurred in the power supply to the encoder (5V/24V) or D/A converter (+/-15V).
Reaction:	NC not ready. The NC switches to follow-up mode. Mode group not ready, also effective for single axes. NC Start disable in this channel. Axes of this channel must be re-referenced. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check the encoder and cable for short-circuits (the fault should not occur when you remove the cable). Check the power feeder line.
Program Continuation:	Switch control OFF - ON.

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

Definitions:	The initialization switch is currently set to overall reset. This means that the module's SRAM is deleted with the next module reset. The NC data memory is cleared during this operation.
Reaction:	NC not ready. Interface signals are set. Alarm display.
Remedy:	Reset initialization switch to zero.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

2900   Reboot is delayed

Definitions:	This alarm indicates a delayed reboot. This alarm only occurs when reboot was carried out by the HMI and MD10088 $MN_REBOOT_DELAY_TIME was set greater than zero. The alarm can be suppressed with MD11410 $MN_SUPPRESS_ALARM_MASK Bit 20.
Reaction:	NC not ready. The NC switches to follow-up mode. Mode group not ready, also effective for single axes. Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Alarm reaction delay is cancelled.
Remedy:	See MD10088 $MN_REBOOT_DELAY_TIME and MD11410 $MN_SUPPRESS_ALARM_MASK.
Program Continuation:	Switch control OFF - ON.

3000   Emergency stop

Definitions:	The EMERGENCY STOP request is applied to the NCK/PLC interface DB2600 DBX0.1 (Emergency stop).
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Alarm reaction delay is cancelled.

Remedy:	Please inform the authorized personnel/service department. Remove the cause of the emergency stop and acknowledge the emergency stop via the PLC/NCK interface DB2600 DBX0.2 (emergency stop acknowledgement).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

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

Parameters:	%1 = Channel number %2 = String: MD identifier
Definitions:	The assignment of a machine axis to a channel by the MD20070 $MC_AXCONF_MACHAX_USED must be contiguous. At system power-up (Power On) gaps are detected and displayed as an alarm.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. The entries for the indices for the machine axes used in the channels must be contiguous in table MD20070 $MC_AXCONF_MACHAX_USED. Channel axis gaps must be enabled via MD11640$MN_ENABLE_CHAN_AX_GAP.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number %2 = String: MD identifier %3 = Index: MD array index
Definitions:	Only axes that have been activated in the channel by means of MD20070 $MC_AXCONF_MACHAX_USED [kx]=m may be declared as geometry axes or transformation axes by means of the MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB [gx]=k. This also applies to MD22420 $MC_FGROUP_DEFAULT_AXES (gx: Geometry axis index, kx: Channel axis index, k: Channel axis no., m: Machine axis no.). Assignment of geometry axes to channel axes MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB (includes channel axis no. k): - Geometry axis index: 0, 1st channel: 1, 2nd channel: 1 - Geometry axis index: 1, 1st channel: 2, 2nd channel: 0 - Geometry axis index: 2, 1st channel: 3, 2nd channel: 3 MD20070 $MC_AXCONF_MACHAX_USED (includes machine axis no. m): - Channel axis index: 0, 1st channel: 1, 2nd channel: 4 - Channel axis index: 1, 1st channel: 2, 2nd channel: 5 - Channel axis index: 2, 1st channel: 3, 2nd channel: 6 - Channel axis index: 3, 1st channel: 7, 2nd channel: 0 - Channel axis index: 4, 1st channel: 8, 2nd channel: 0 - Channel axis index: 5, 1st channel: 0, 2nd channel: 0 - Channel axis index: 6, 1st channel: 0, 2nd channel: 0 - Channel axis index: 7, 1st channel: 0, 2nd channel: 0
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct - MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB - MD24... $MC_TRAFO_AXES_IN_... - MD24... $MC_TRAFO_GEOAX_ASSIGN_TAB_... - MD22420 $MC_FGROUP_DEFAULT_AXES - and/or MD20070 $MC_AXCONF_MACHAX_USED.
Program Continuation:	Switch control OFF - ON.

4004	[Channel %1:] Machine data %2 axis %3 defined repeatedly as geometry axis
Parameters:	%1 = Channel number %2 = String: MD identifier %3 = Axis index
Definitions:	An axis may only be defined once as a geometry axis.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Correct MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.
Program Continuation:	Switch control OFF - ON.

4005	[Channel %1:] Maximum number of axes is exceeded. Limit %2
Parameters:	%1 = Channel number %2 = Upper limit for the number of axes in the channel
Definitions:	MD20070 $MC_AXCONF_MACHAX_USED defines which machine axes can be used in this channel. This simultaneously defines the number of active axes in the channel. This upper limit has been exceeded. Note: The channel axis gaps may cause certain indices of MD20070 $MC_AXCONF_MACHAX_USED to remain unused and therefore do not count as active channel axes. Example: - CHANDATA(2) - $MC_AXCONF_MACHAX_USED[0] = 7 - $MC_AXCONF_MACHAX_USED[1] = 8 - $MC_AXCONF_MACHAX_USED[2] = 0 - $MC_AXCONF_MACHAX_USED[3] = 3 - $MC_AXCONF_MACHAX_USED[4] = 2 - $MC_AXCONF_MACHAX_USED[5] = 0 - $MC_AXCONF_MACHAX_USED[6] = 1 - $MC_AXCONF_MACHAX_USED[7] = 0 This channel uses the five machine axes 1, 2, 3, 8, 7, i.e. it has 5 active channel axes.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify MD20070 $MC_AXCONF_MACHAX_USED.
Program Continuation:	Switch control OFF - ON.

4006	The maximum number of activatable axes has been exceeded (limit %1)
Parameters:	%1 = Number of axes
Definitions:	The sum of the two option data $ON_NUM_AXES_IN_SYSTEM and $ON_NUM_ADD_AXES_IN_SYSTEM must not exceed the maximum number of axes in the system.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personel/service department. The sum of the two option data $ON_NUM_AXES_IN_SYSTEM and $ON_NUM_ADD_AXES_IN_SYSTEM must not exceed the maximum number of axes (dependent on configuration).

Program Continuation:	Switch control OFF - ON.

4009	**Machine data %1 contains an illegal value.**
Parameters:	%1 = String: MD identifier
Definitions:	A value has been entered which exceeds the value range or a limit value for a variable, a machine data or a function.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Enter correct values.
Program Continuation:	Switch control OFF - ON.

4010	**Invalid identifier used in machine data %1[%2]**
Parameters:	%1 = String: MD identifier %2 = Index: MD array index
Definitions:	When determining a name in the NCK tables (arrays) for: machine axes, Euler angles, direction vectors, normal vectors, interpolation parameters and intermediate point coordinates, one of the following syntax rules for the identifier to be entered has been violated: - The identifier must be an NC address letter (A, B, C, I, J, K, Q, U, V, W, X, Y, Z), possibly with a numerical extension (840D: 1-99) - The identifier must begin with any 2 capital letters but not with $ (reserved for system variables). - The identifier must not be a keyword of the NC language (e.g. POSA).
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Enter the identifier for user-defined names with correct syntax in the displayed MD. - Machine axes: MD10000 $MN_AXCONF_MACHAX_NAME_TAB - Euler angles: MD10620 $MN_EULER_ANGLE_NAME_TAB - Normal vectors: MD10630 $MN_NORMAL_VECTOR_NAME_TAB - Direction vectors: MD10640 $MN_DIR_VECTOR_NAME_TAB - Interpolation parameters: MD10650 $MN_IPO_PARAM_NAME_TAB - Intermediate point coordinates: MD10660 $MN_INTERMEDIATE_POINT_NAME_TAB
Program Continuation:	Switch control OFF - ON.

4011	**[Channel %1:] Invalid identifier used in machine data %2[%3]**
Parameters:	%1 = Channel number %2 = String: MD identifier %3 = Index: MD array index
Definitions:	When defining names in the channel-specific tables for geometry axes and channel axes, one of the following syntax rules for the identifier to be entered has been violated: - The identifier must be an NC address letter (A, B, C, I, J, K, U, V, W, X, Y, Z), possibly with a numerical extension. - The identifier must begin with any 2 capital letters but not with $ (reserved for system variables). - The identifier must not be a keyword of the NC language (e.g. SPOS).
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.

Remedy:	Please inform the authorized personnel/service department.
	Enter the identifier for user-defined names with correct syntax in the displayed MD
	- Geometry axes: MD20060 $MC_AXCONF_GEOAX_NAME_TAB
	- Channel axes: MD20080 $MC_AXCONF_CHANAX_NAME_TAB
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = String: MD identifier
	%2 = Index: MD array
Definitions:	The selected identifier is invalid. Valid identifiers are:
	- AX1 - AXn: Machine axis identifiers
Reaction:	NC not ready.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Use the correct identifier.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = String: Name of identifier
	%2 = String: MD identifier
Definitions:	When determining a name in the NCK tables (arrays) for: machine axes, Euler angles, direction vectors, normal vectors, interpolation parameters and intermediate point coordinates, an identifier has been used that already exists in the control.
Reaction:	NC not ready.
	Mode group not ready, also effective for single axes.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Select for the identifier to be entered a character string that is not yet used in the system (max. 32 characters).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

4021   [Channel %1:] Identifier %2 used several times in machine data %3

Parameters:	%1 = Channel number
	%2 = String: Name of identifier
	%3 = String: MD identifier
Definitions:	To determine the name in the channel-specific tables for geometry axes and channel axes an identifier already existing in the control has been used.
Reaction:	NC not ready.
	Mode group not ready, also effective for single axes.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Select for the identifier to be entered a character string that is not yet used in the system (max. 32 characters).
Program Continuation:	Switch control OFF - ON.

4030	[Channel %1:] Identifier missing in machine date %2[%3]
Parameters:	%1 = Channel number %2 = String: MD identifier %3 = Index: MD array index
Definitions:	An axis identifier is expected for the displayed MD in accordance with the axis configuration in the MD20070 $MC_AXCONF_MACHAX_USED and MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check axis configuration and enter the missing identifier into the MD or, should the axis not exist, specify for this channel axis the machine axis 0 in the channel-specific MD20070 $MC_AXCONF_MACHAX_USED. If this concerns a geometry axis that is not to be used (this applies only for 2-axis machining, e.g. on lathes), then channel axis 0 must be entered additionally in the channel-specific MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.
Program Continuation:	Switch control OFF - ON.

4032	[Channel %1:] Wrong identifier for facing axis in %2
Parameters:	%1 = Channel number %2 = String: MD identifier
Definitions:	According to the axis configuration in MD20150 $MC_GCODE_RESET_VALUES or MD20100 $MC_DIAMETER_AX_DEF, a facing axis identifier is expected at the specified location.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Add the correct identifier.
Program Continuation:	Switch control OFF - ON.

4040	[Channel %1:] Axis identifier %2 not consistent with machine data %3
Parameters:	%1 = Channel number %2 = String: Axis identifier %3 = String: MD identifier %4 = There are not enough channel axes entered in the MD displayed.
Definitions:	The use of the specified axis identifier in the displayed MD is not consistent the channel's axis configuration stated in the MD20070 $MC_AXCONF_MACHAX_USED and MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check and correct the identifier used in the MD10000 $MN_AXCONF_MACHAX_NAME_TAB, MD20080 $MC_AXCONF_CHANAX_NAME_TAB and/or MD20060 $MC_AXCONF_GEOAX_NAME_TAB.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number %2 = String: MD identifier %3 = String: MD identifier
Definitions:	Using the specified machine data %1 leads to a conflict with machine data %2.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Correct the specified machine data.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = String: Old identifier %2 = String: New identifier
Definitions:	Renaming of an NC code was not possible for one of the following reasons: - The old identifier does not exist - The new identifier lies in another type range. NC codes/keywords can be reconfigured via machine data as long as the type range is not abandoned. Type 1: "real" G codes: G02, G17, G33, G64, ... Type 2: named G codes: ASPLINE, BRISK, TRANS, ... Type 3: settable addresses: X, Y, A1, A2, I, J, K, ALF, MEAS, ...
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct MD10712 $MN_NC_USER_CODE_CONF_NAME_TAB (protection level 1). The list must be built up as follows: Even address: Identifier to be modified. Following odd address: New identifier e.g.: $MN_NC_USER_CODE_CONF_NAME_TAB [10] = "ROT" $MN_NC_USER_CODE_CONF_NAME_TAB [11] = " " clears the ROT function from the control
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Identifier 1 %2 = Identifier 2 %3 = Identifier 3 %4 = Identifier 4
Definitions:	The standard MD were loaded because - a cold start was requested or - the MD buffer voltage failed or - an initialization was requested for loading the standard machine data (MD11200 $MN_INIT_MD).
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. After automatically loading the standard MDs, the individual MDs must be entered or loaded in the relevant system.
Program Continuation:	Clear alarm with the RESET key. Restart part program

4062	Backup data loaded
Definitions:	The user data saved in the flash memory are loaded to the SRAM.
Reaction:	Alarm display.
Remedy:	Load specific machine data again.
Program Continuation:	Clear alarm with the RESET key. Restart part program

4065	Buffered memory was restored from backup medium (potential loss of data!)
Definitions:	Only occurs with SINUMERIK 840D / 840Di sl / 802D. !! 840Di sl only The user data of the NC and the remanent data of the PLC are stored in the static memory area (SRAM) of the MCI board. The content of the SRAM is backed up as an SRAM image on PCU hard disk at each "NCK POWER ON reset" and each time Windows XP is closed down normally. The previously valid SRAM image then becomes the SRAM backup, which is also stored on the PCU hard disk. The SRAM backup is used and alarm 4065 issued in the following cases:

HW serial no MCI board	SRAM MCI board "OK"	SRAM image "OK"
1. Known	No	No
2. Unknown	Yes	No
3. Unknown	No	No

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

Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Make a POWER ON reset. !! 840Di / 840Di sl only: Alarm 4065 also has to be acknowledged on the HMI after a POWER ON reset: HMI: Operating area switchover > Diagnostics > NC/PLC Diagnostics > Diagnostics > "Acknowledge alarm 4065" button Note Press the "ETC" key to change to the secondary softkey bar in order to acknowledge the alarm with a softkey.
Program Continuation:	Switch control OFF - ON.

4066	Buffered memory of FFS restored from backup medium (potential loss of data!)
Definitions:	For 840Di: A possible data integrity error was detected in the FFS memory during power-up. The FFS memory was initialized with the last backup copy. Changes in the FFS memory, which have been made since the last backup copy update, have been lost. !! For 840Di only: Backup copies of the buffered memory are updated (on the hard disk) every time the control is shut down normally.
Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Start the control again.
Program Continuation:	Switch control OFF - ON.

4070   Normalizing machine data has been changed

Definitions:	The control uses internal physical units (mm, degrees, s, for paths, velocities, acceleration, etc.). During programming or data storage, some of these values are input and output using different units (rev./min, m/s2, etc.).
	The conversion is carried out with the scaling factors that can be entered (system-spefic MD array MD10230 $MN_SCALING_FACTORS_USER_DEF USER_DEF[n] (n ... index number 0 - 10), when the corresponding masking bit is set to "1".
	If the masking bit is set to "0" then scaling takes place with the internal standard factors.
	The following machine data influence the scaling of other MDs:
	MD10220: $MN_SCALING_USER_DEF_MASK
	MD10230: $MN_SCALING_FACTORS_USER_DEF
	MD10240: $MN_SCALING_SYSTEM_IS_METRIC
	MD10250: $MN_SCALING_VALUE_INCH
	MD30300: $MA_IS_ROT_AX
	If these data are modified, the NCK must be powered up again. Only then will the input of dependent data be performed correctly.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department.
	If the alarm has been displayed after downloading an MD file which is consistent within itself, then the download operation must be repeated with a new NC power-up. (The file contains scaling-dependent machine data in front of the scaling factors).
Program Continuation:	Clear alarm with the Delete key or NC START.

4071   Check the position of the encoder

Definitions:	A machine data has been changed that affects the value of an absolute encoder position. Please check the postion values.
	For absolute encoders:
	Encoder adjustment has been changed, the machine reference of the axis position may have changed, check the encoder adjustment.
	Other encoders:
	The reference point of the axis position has been changed, check the referencing procedure.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department.
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Parameters:	%1 = String: MD identifier
	%2 = Write protection level of the MD
Definitions:	On executing a TOA file or when writing machine data from the part program, an attempt has been made to write an item of data with a higher protection level than the access authorization currently set in the control. The item of data in question has not been written and program execution is continued. This alarm is set only when access violation is detected for the first time.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Set the required access level by means of keyswitch or password entry or delete the machine data concerned from the MD file/part program.
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Parameters:	%1 = Number of MDs
	%2 = Preset access authorization
Definitions:	On executing a TOA file or when writing data from the part program an attempt has been made to write data with a higher protection level than the access authorization currently set in the control. The data in question have not been written and program execution is continued without hindrance. This alarm is issued on acknowledging alarm 4075. It can be cleared only with Power ON.

Reaction:	NC Start disable in this channel. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Set the required access level by means of keyswitch or password entry or delete the machine data concerned from the MD file/part program.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = New value of machine data %2 = Machine data number %3 = Number of bytes requested that exceeded availability %4 = Type of memory
Definitions:	An attempt was made to enter a new value in the specified memory configuration machine data. It was not possible to modify the value, as this would have cleared the contents of the user memory. This was because the memory requested exceeded the available capacity. The third parameter specifies the number of bytes by which the maximum user memory was exceeded. The fourth parameter specifies the type of memory whose limit was exceeded. - "D" stands for dynamic or non-buffered user memory (this is where, for example, the LUD variables are stored and the interpolation buffer size is entered). The capacity of this memory type is defined by the current memory configuration and the value in MD18210 $MN_MM_USER_MEM_DYNAMIC. - "S" stands for static or buffered user memory (this is where part programs, offset data, R parameters, tool data, etc. are stored). This memory type is defined by the current memory configuration and the value in MD18230 $MN_MM_USER_MEM_BUFFERED. - "iS" stands for internal static or buffered user memory. This memory type is defined by the current memory configuration (not settable). A few NCK functions use this memory.
Reaction:	Alarm display.
Remedy:	If the modification was unintentional, ignore the error message and continue. The alarm has no negative effects. The remedy depends on the access right and the current memory configuration of the NCK: - The intended change is not possible -> Try again with a smaller value. Observe the change in the number of bytes. - Is it possible to expand the memory? This option depends on the model in use. (Not possible if parameter 4 equals "iS"). - The NCK user memory may have been set smaller than it could be. With the appropriate access authorization, the machine data (see above) can be changed. - If parameter 4 equals "iS" and no synchronous actions are used, then MD18232 $MN_MM_ACTFILESYS_LOG_FILE_MEM[2] = 0 can be set. Otherwise the desired machine data change cannot be made.
Program Continuation:	Clear alarm with the Delete key or NC START.

4080   Incorrect configuration of indexing axis in MD %1

Parameters:	%1 = String: MD identifier
Definitions:	The assignment of a position table to an indexing axis or the contents of a position table contains an error, or the length of a position table has been parameterized with 0.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. 3 MD identifiers are output, depending on the type of error. 1. MD30500 $MA_INDEX_AX_ASSIGN_POS_TAB: the error is due to multiple assignment of a position table MD10910 $MN_INDEX_AX_POS_TAB_1 or MD10930 $MN_INDEX_AX_POS_TAB_2) to axes with different types (linear/rotary axis). 2. MD10910 $MN_INDEX_AX_POS_TAB_1 or MD10930 $MN_INDEX_AX_POS_TAB_2: the contents of the displayed tables are incorrect. - The entered positions must be arranged in increasing size. - A particular position must not be set more than once. - If the table is assigned to one or several modulo axes, then the contents must be within the 0 to < 360 degree range.

3. MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1 or MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2: the length of the displayed position table n was specified with 0.

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

---

4082   [channel %1:] invalid value in machine date %2%3

Parameters:	%1 = Channel number %2 = String: MD identifier %3 = String: MD field index
Definitions:	A value has been entered which exceeds the value range or a limit value for a variable, a machine data or a function.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Enter correct values.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

4090   Too many errors during power-up

Definitions:	More than <n> errors occurred during control power-up.
Reaction:	NC Start disable in this channel. Alarm display.
Remedy:	Set the machine data correctly.
Program Continuation:	Switch control OFF - ON.

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

Definitions:	MD10050 $MN_SYSCLOCK_CYCLE_TIME (system clock cycle) and/or MD10080 $MN_SYSCLOCK_SAMPL_TIME_RATIO (division factor of the position control cycle for actual value acquisition) have been corrected. The new value of the system clock cycle can be taken from MD10050 $MN_SYSCLOCK_CYCLE_TIME.
Reaction:	Alarm display.
Remedy:	No remedial measures are required. The alarm display can be canceled with Reset.
Program Continuation:	Clear alarm with the Delete key or NC START.

4110   IPO cycle changed to %1 ms

Parameters:	%1 = String (new IPO cycle time)
Definitions:	The IPO cycle divisor was set to a value which was not an integral multiple of the position control cycle divisor. The divisor (MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO) was increased.
Reaction:	Alarm display.
Remedy:	MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO has been modified.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

4111   PLC cycle increased to %1 ms

Parameters:	%1 = String (new PLC cycle time)
Definitions:	The PLC cycle divisor was set to a value which was not an integral multiple of the IPO cycle divisor. The divisor (MD10074 $PLC_IPO_TIME_RATIO) has been increased.
Reaction:	Alarm display.
Remedy:	MD10074 $MN_PLC_IPO_TIME_RATIO has been modified.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

4112 **Servo cycle changed to %1 ms**

Parameters:	%1 = String (new servo cycle time)
Definitions:	
Reaction:	Alarm display.
Remedy:	MD10060 $MN_POSCTRL_SYSCLOCK_TIME_RATIO RATIO has been modified.
Program Continuation:	Clear alarm with the RESET key. Restart part program

4113 **Sysclock cycle changed to %1 ms**

Parameters:	%1 = String (new PLC cycle time)
Definitions:	
Reaction:	Alarm display.
Remedy:	MD10050 $MN_SYSCLOCK_CYCLE_TIME has been modified.
Program Continuation:	Clear alarm with the RESET key. Restart part program

4115 **Time ratio communication to Ipo changed to %1**

Parameters:	%1 = String (new PLC cycle time)
Definitions:	The value of the MD10072 $MN_COM_IPO_TIME_RATIO has been adapted. This can only occur, if the value of the machine data is smaller than one and the time thus calculated is no multiple of the position control cycle.
Reaction:	Alarm display.
Remedy:	The MD10072 $MN_COM_IPO_TIME_RATIO has been adapted. Please check to ensure that the calculated value is correct.
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number
Definitions:	MD10715 $MN_M_NO_FCT_CYCLE[n] or MD10718 $MN_M_NO_FCT_CYCLE_PAR contains invalid configuration data: An M function, which is occupied by the system and cannot be replaced by a subprogram call has been specified in MD10715 $MN_M_NO_FCT_CYCLE[n] for the configuration of the subprogram call via M function: - M0 to M5, - M17, M30, - M19, M40 to M45, - M function for selecting spindle/axis mode according to MD20094 $MC_SPIND_RIGID_TAPPING_M_NR (default: M70), - M functions for nibbling/punching as configured in MD26008 $MC_NIBBLE_PUNCH_CODE if activated by MD26012 $MC_PUNCHNIB_ACTIVATION. - Also M96 to M99 for applied external language (MD18800 $MN_MM_EXTERN_LANGUAGE). MD10718 $MN_M_NO_FCT_CYCLE_PAR contains an invalid array index of MD10715 $MN_M_NO_FCT_CYCLE[n]. Currently, the values 0 to 9 are permissible. The affected machine data is reset to the default value -1. This deactivates the function.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Configure an M function in MD10715 $MN_M_NO_FCT_CYCLE[n] that is not occupied by the system, or configure a permissible array index in MD10718 $MN_M_NO_FCT_CYCLE_PAR.
Program Continuation:	Switch control OFF - ON.

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

Definitions:	The "Block display with absolute values" function has been illegally parameterized:
	- An illegal block length has been set with MD28400 $MC_MM_ABSBLOCK:
	While ramping up, the machine data will be checked for the following value range:
	0, 1, 128 to 512
	- An invalid display range has been set with MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[]. While ramping up, the machine data will be checked for the following upper and lower limits:
	   0 <= MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[0] <= 8
	   0 <= MD28402 $MC_MM_ABSBLOCK_BUFFER_CONF[1] <= (MD28060 $MC_MM_IPO_BUFFER_SIZE + MD28070$MC_MM_NUM_BLOCKS_IN_PREP).
	Alarm 4152 is issued if the limits are violated.

Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Configure block length/display range within the permissible limits.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number
Definitions:	An M function was specified in MD20094 $MC_SPIND_RIGID_TAPPING_M_NR in order to configure the M function number for spindle switchover. The M function number is assigned by the system and cannot be used for the switchover (M1 to M5, M17, M30, M40 to M45).
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Configure an M function which is not used by the system (M1 to M5, M17, M30, M40 to M45) in MD20094 $MC_SPIND_RIGID_TAPPING_M_NR.
Program Continuation:	Switch control OFF - ON.

4180 Invalid M function number assigned to enable ASUP

Definitions:	An invalid M function number has been assigned for activation of ASUP. An illegal M number has been assigned in MD10804 $MN_EXTERN_M_NO_SET_INT or MD10806 $MN_EXTERN_M_NO_DISABLE_INT for the configuration of the M number range for activation/deactivation of the interrupt program.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Check MD10804 $MN_EXTERN_M_NO_SET_INT and MD10806 $MN_EXTERN_M_NO_DISABLE_INT.
Program Continuation:	Switch control OFF - ON.

4181 [Channel %1:] Invalid assignment of an M auxiliary function number

Parameters:	%1 = Channel number
Definitions:	In MD22254 $MC_AUXFU_ASSOC_M0_VALUE or MD22256 $MC_AUXFU_ASSOC_M1_VALUE, a number has been specified for the configuration of a new predefined M function which is occupied by the system, and cannot be used for an assignment. (M0 to M5, M17, M30, M40 to M45).

Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Configure an M function in machine data MD22254 $MC_AUXFU_ASSOC_M0_VALUE or MD22256 $MC_AUXFU_ASSOC_M1_VALUE which is not occupied by the system (M1 to M5, M17, M30, M40 to M45).
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number %2 = MD identifier %3 = If required, MD index
Definitions:	In the specified machine data, a number has been specified for the configuration of an M function which is occupied by the system, and cannot be used for an assignment. (M0 to M5, M17, M30, M40 to M45 and also M98, M99 with applied ISO dialect). The value set by the user has been reset to the default value by the system.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Configure an M function in the specified machine data which is not occupied by the system (M0 to M5, M17, M30, M40 to M45 and also M98, M99 with applied ISO dialect).
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel number %2 = M auxiliary function number %3 = MD identifier %4 = MD identifier
Definitions:	In the specified machine data, a number has been used several times for the configuration of an M function.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Check the specified machine data and create a unique assignment of M auxiliary function numbers.
Program Continuation:	Switch control OFF - ON.

4184 [Channel %1:] Illegally predefined auxiliary function in %2[%3], MD reset

Parameters:	%1 = Channel number %2 = MD identifier %3 = If required, MD index
Definitions:	In the specified machine data, a predefined auxiliary function has been illegally configured. The value set by the user has been reset to the default value by the system.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.

Remedy:	Configure a valid value in the specified machine data.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

4185   [Channel %1:] Illegal auxiliary function configured %2 %3 %4

Parameters:	%1 = Channel number %2 = Type of auxiliary function %3 = Extension %4 = Auxiliary function value
Definitions:	An auxiliary function has been illegally configured. Predefined auxiliary functions cannot be reconfigured by user-defined auxiliary functions. See: MD22010 $MC_AUXFU_ASSIGN_TYPE[n] MD22020 $MC_AUXFU_ASSIGN_EXTENSION[n] MD22030 $MC_AUXFU_ASSIGN_VALUE[n] MD22035 $MC_AUXFU_ASSIGN_SPEC[n]
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Reconfigure the auxiliary function.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

4200   [Channel %1:] Geometry axis %2 must not be declared a rotary axis

Parameters:	%1 = Channel number %2 = Axis name
Definitions:	The geometry axes represent a Cartesian coordinate system and therefore the declaration of a geometry axis as rotary axis leads to a definition conflict.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Remove rotary axis declaration for this machine axis. For this purpose, the geometry axis index for the displayed geometry axis must be determined by means of MD20060 $MC_AXCONF_GEOAX_NAME_TAB. The channel axis number is stored with the same index in MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB. The channel axis number minus 1 provides the channel axis index under which the machine axis number is found in MD20070 $MC_AXCONF_MACHAX_USED.
Program Continuation:	Switch control OFF - ON.

4210   [Channel %1:] Spindle %2 declaration as rotary axis missing

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	If a machine axis is to be operated as a spindle, this machine axis must be declared as a rotary axis.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.

Remedy:	Please inform the authorized personnel/service department. Set rotary axis declaration for this machine axis in the axis-specific MD30300 $MA_IS_ROT_AX.
Program Continuation:	Switch control OFF - ON.

4215 [Channel %1:] Spindle %2 declaration as modulo axis missing

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The spindle functionality requires a modulo axis (positions in [deg]).
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Set MD30310 $MA_ROT_IS_MODULO.
Program Continuation:	Switch control OFF - ON.

4220 [Channel %1:] Spindle %2 declared repeatedly

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The spindle number exists more than once in the channel.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. The spindle number is stored in the axis-specific MD35000 $MA_SPIND_ASSIGN_TO_MACHAX. The channel to which this machine axis/spindle is assigned is listed in the machine axis index. (The machine axis number is given in the channel-specific MD20070 $MC_AXCONF_MACHAX_USED).
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number %2 = Axis name, axis number
Definitions:	The modulo functionality requires a rotary axis (positions in [deg]).
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Set MD30300 $MA_IS_ROT_AX.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Channel number

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

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

Parameters:	%1 = Program location
Definitions:	The settings for the interpolation and position control cycle were modified before the last power-up such that too little computing time is now available for the requisite cyclic task. The alarm occurs immediately after power-up if too little runtime is available even when the axes are stationary and the NC program has not started. However, task overflow can occur only when computation-intensive NC functions are called during program execution.
Reaction:	NC not ready. The NC switches to follow-up mode. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Alarm reaction delay is cancelled.
Remedy:	Please inform the authorized personnel/service department. Take greater care when optimizing the clock times MD10050 $MN_SYSCLOCK_CYCLE_TIME, MD10060 $MN_POSCTRL_SYSCLOCK_TIME_RATIO and/or MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO. The test should be performed with an NC program that represents the highest possible control load. To be on the safe side, a margin of 15 to 25% should be added to the times determined in this way.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = String: MD identifier %2 = Index
Definitions:	The specified machine data assigns a digital input/output byte or an analog input/output signal the processing of which has not been activated to an NC function.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct machine data. Activate required inputs/outputs via MDs: MD10350 $MN_FASTIO_DIG_NUM_INPUTS MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS MD10300 $MN_FASTIO_ANA_NUM_INPUTS MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS Activation of fast inputs/outputs does not require the corresponding hardware configuration to be available at the control. All functions using fast inputs/outputs can also be made use of by the PLC specification/modification defined in the VDI interface, if the response time requirements are reduced accordingly. Activated inputs/outputs increase the computation time requirement of the interpolation cycle because the PLC manipulation signals are handled cyclically. Note: Deactivate any inputs/outputs not in use.
Program Continuation:	Switch control OFF - ON.

4275	**Machine data %1 and %2 both assign the same NCK output byte no. %3 several times**
Parameters:	%1 = String: MD identifier %2 = String: MD identifier %3 = No. of output
Definitions:	The specified machine data assign two NC functions to the same digital/analog output.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct machine data.
Program Continuation:	Switch control OFF - ON.

4280	**Assignment of NCK input/output byte via MD %1[%2] does not match hardware configuration**
Parameters:	%1 = String: MD identifier %2 = Index: MD array
Definitions:	When booting, the required input/output module was not found at the slot specified in the MD.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check hardware or correct the MD, if necessary. Note: monitoring of the hardware configuration is performed independently of the number of activated inputs/outputs (MD10300 $MN_FASTIO_ANA_NUM_INPUTS, MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS, MD10350 $MN_FASTIO_DIG_NUM_INPUTS, MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS)
Program Continuation:	Switch control OFF - ON.

4282	**Hardware of external NCK outputs assigned repeatedly**
Definitions:	Several outputs have been configured on the same hardware byte.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Alter MD10368 $MN_HW_ASSIGN_DIG_FASTOUT or MD10364 $MN_HW_ASSIGN_ANA_FASTOUT.
Program Continuation:	Switch control OFF - ON.

4300	**Declaration in MD %1 is not allowed for axis %2.**
Parameters:	%1 = String: MD identifier %2 = Axis name, spindle number
Definitions:	The axis cannot be operated as competing positioning axes, for example because the axis is the slave axis within a closed gantry group or a gantry group to be closed.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Reset MD30450 $MA_IS_CONCURRENT_POS_AX for the axis concerned.

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

---

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

Parameters:	%1 = String: MD identifier
	%2 = Index: MD array index
Definitions:	The machine data values must be written in the array in ascending order.
Reaction:	Mode group not ready.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct the MD.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

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

Parameters:	%1 = String: Axis identifier
	%2 = String: MD identifier
	%3 = String: Bit
	%4 = String: MD identifier
Definitions:	The functions declared by the specified machine data cannot simultaneously be active for one axis.
Reaction:	Mode group not ready.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Deactivate one of the functions.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Memory type
	%2 = MD identifier, if required
Definitions:	A machine data has been altered that configures the buffered memory. If the NCK powers up with the altered data, this will lead to reorganization of the buffered memory and thus to the loss of all buffered user data (part programs, tool data, GUD, leadscrew error compensation, ...)
	Meaning of the 1st parameter
	0x00 buffered memory (internal)
	0x01 buffered memory
Reaction:	Alarm display.
Remedy:	If the control includes user data that have not yet been saved, then a data backup must be performed before the next NC power-up. By manually resetting the altered MD to the value it had before the last power-up, reorganization of the memory can be avoided.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

4402   %1 causes a machine data reset

Parameters:	%1 = Machine data
Definitions:	If this machine data is set, the current machine data values are overwritten by the default values at the next ramp-up. Under certain circumstances, this may cause data loss (even in the buffered memory).
Reaction:	Alarm display.

Remedy:	Please inform the authorized personnel/service department. If the control includes user data that has not yet been saved, then a data backup must be performed before the next NCK power-up. By manually resetting the altered MD to the value it had before the last power-up, reorganization of the memory can be avoided.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

4502 [Channel %1:] Anachronism %2(%3) -> %4

Parameters:	%1 = Channel number %2 = String: MD identifier %3 = String: MD identifier %4 = String: MD identifier
Definitions:	Previously, in MD20110 $MC_RESET_MODE_MASK Bit4 and Bit5, the reset behavior of the 6th or 8th G groupe was determined. This setting is now made in MD20152 $MC_GCODE_RESET_MODE. In order to ensure compatible handling of "old" data backups, the "old" values are taken from MD20110 $MC_RESET_MODE_MASK and entered in MD20152 $MC_GCODE_RESET_MODE.
Reaction:	Alarm display.
Remedy:	--
Program Continuation:	Clear alarm with the Delete key or NC START.

4503 [TO unit %1:] H number %2 assigned more than once. Machine data is not set.

Parameters:	%1 = TO unit %2 = H number
Definitions:	This error can only occur if MD10880 $MN_MM_EXTERN_CNC_SYSTEM= 1 or 2. MD10890, $MN_EXTERN_TOOLPROG_MODE bit 3 is reset (this MD becomes effective at power-on). On checking data management, it was found that various edges of the same TO unit had the same H number. MD10890 $MN_EXTERN_TOOLPROG_MODE bit 3 remains set and is not included in data management.
Reaction:	Alarm display.
Remedy:	H numbers must be assigned only once in a TO unit. Then, MD10890, $MN_EXTERN_TOOLPROG_MODE, bit 3 can be set = 0 and a restart can be performed.
Program Continuation:	Clear alarm with the Delete key or NC START.

4600 Invalid handwheel type for handwheel %1

Parameters:	%1 = Handwheel number
Definitions:	The handwheel type (hardware segment) for handwheel %1 requested through MD11350 $MN_HANDWHEEL_SEGMENT is invalid.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Configure a valid type for the corresponding handwheel through MD11350 $MN_HANDWHEEL_SEGMENT.
Program Continuation:	Switch control OFF - ON.

4610 Invalid handwheel module for handwheel %1

Parameters:	%1 = Handwheel module
Definitions:	For SINUMERIK 840D and SINUMERIK 840DI only: The handwheel module for handwheel %1 requested through MD11351 $MN_HANDWHEEL_MODULE is not available for 840D systems. An 840D system is always regarded as a module. Therefore MD11351 $MN_HANDWHEEL_MODULE = 1 must always be set for handwheels directly linked to 840D systems.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Set machine date MD11351 $MN_HANDWHEEL_MODULE = 1 for the corresponding handwheel.
Program Continuation:	Switch control OFF - ON.

4611	**Invalid handwheel input for handwheel %1**
Parameters:	%1 = Handwheel input
Definitions:	For SINUMERIK 840D and SINUMERIK 840Di only:
	The handwheel input for handwheel %1 requested through MD11352 $MN_HANDWHEEL_INPUT is not available for 840D systems. A maximum of 2 or 3 handwheels can be linked directly to 840D systems:
	840D powerline: 1st and 2nd handwheels directly to the 840D hardware
	840Di: 1st and 2nd handwheels directly to the extension board
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Configure MD11352 $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel
Program Continuation:	Switch control OFF - ON.

4620	**Invalid handwheel module for handwheel %1**
Parameters:	%1 = Handwheel module
Definitions:	The handwheel module for handwheel %1 requested through MD11351 $MN_HANDWHEEL_MODULE is not available for 802D sl, 828D sl, 808D systems. 802D sl, 828D sl, 808D systems are always regarded as modules. Therefore MD11351 $MN_HANDWHEEL_MODULE = 1 must always be set for directly linked handwheels.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Set machine data MD11351 $MN_HANDWHEEL_MODULE = 1 for the corresponding handwheel.
	For 840D sl systems, MD11350 $MN_HANDWHEEL_SEGMENT should be checked.
Program Continuation:	Switch control OFF - ON.

4621	**Invalid handwheel input for handwheel %1**
Parameters:	%1 = Handwheel input
Definitions:	The handwheel input for handwheel %1 requested through MD11352 $MN_HANDWHEEL_INPUT is not available for 802D sl, 828D sl, 808D systems. A maximum of 2 handwheels can be directly linked to 802D sl, 828D sl, 808D systems.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Configure MD11352 $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel.
	For 840D sl systems, MD11350 $MN_HANDWHEEL_SEGMENT should be checked.
Program Continuation:	Switch control OFF - ON.

4630	**Invalid handwheel module for handwheel %1**
Parameters:	%1 = Handwheel module
Definitions:	
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Configure the machine data MD11351 $MN_HANDWHEEL_MODULE for the corresponding PROFIBUS handwheel so that there is a valid reference to an entry in the machine data array MD11353 $MN_HANDWHEEL_LOGIC_ADDRESS[].
Program Continuation:	Switch control OFF - ON.

4631	**Invalid handwheel slot for handwheel %1**
Parameters:	%1 = Handwheel slot
Definitions:	
Reaction:	Interface signals are set.
	Alarm display.

Remedy:	Configure machine date MD11352 $MN_HANDWHEEL_INPUT to a valid handwheel slot for the corresponding PROFIBUS handwheel.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Handwheel number
Definitions:	
Reaction:	Interface signals are set. Alarm display.
Remedy:	Check if MD11351 $MN_HANDWHEEL_MODULE of the corresponding handwheel is correct. Check if indexed logical base address of PROFIBUS handwheel slot in machine date array MD11353 $MN_HANDWHEEL_LOGIC_ADDRESS[] is correct.
Program Continuation:	Switch control OFF - ON.

4640 Invalid handwheel module for handwheel %1

Parameters:	%1 = Handwheel module
Definitions:	For ETHERNET only: The handwheel module for handwheel %1 requested through MD11351 $MN_HANDWHEEL_MODULE is not available for ETHERNET handwheels. MD11351 $MN_HANDWHEEL_MODULE = 1 must always be set when configuring ETHERNET handwheels.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Set machine date MD11351 $MN_HANDWHEEL_MODULE = 1 for the corresponding handwheel.
Program Continuation:	Switch control OFF - ON.

4641 Invalid handwheel input for handwheel %1

Parameters:	%1 = Handwheel input
Definitions:	For ETHERNET only: The handwheel input for handwheel %1 requested through MD11352 $MN_HANDWHEEL_INPUT is not available for ETHERNET handwheels. A maximum of 6 handwheels can be configured.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Configure MD11352 $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel
Program Continuation:	Switch control OFF - ON.

5000 Communication job not executable %1

Parameters:	%1 = Reference to which resources are no longer available.
Definitions:	The communication job (data exchange between NCK and HMI, e.g.: loading an NC part program) cannot be executed because there is insufficient memory space. Cause: too many communication jobs in parallel.
Reaction:	Alarm display.
Remedy:	- Reduce the number of communication jobs taking place at the same time or increase MD10134 $MN_MM_NUM_MMC_UNITS - Restart communication job. Please inform the authorized personnel/service department. No remedial measures are possible - the operation triggering the alarm message has to be repeated. Clear the alarm display with Cancel.
Program Continuation:	Clear alarm with the Delete key or NC START.

6000   Memory reorganized using standard machine data

Definitions:	The memory management was not able to allocate the NC user memory with the values in the machine data. It did not have enough memory available because the total memory available is provided as dynamic and static memory for the NC user (e.g. for macro definitions, user variables, number of tool offsets, number of directories and files etc.).
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Redefine the NC memory structure! A specific MD for NC user memory allocation cannot be stated to be the cause of the alarm. The MD initiating the alarm therefore has to be determined on the basis of the default values in the machine data by changing the user-specific memory structure step by step. Usually, it is not just one single MD that has been set too large. Therefore it is advisable to reduce the memory area by a certain proportion in several MDs.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

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

Parameters:	%1 = Channel number %2 = String (block name) %3 = Internal error code
Definitions:	Data management has detected an error during ramp-up. The specified data block may not have been created. The error number specifies the type of error. An error number >100000 indicates a fatal system error. Other error numbers indicate that the user memory area provided is too small. In this case the (user) error numbers have the following meaning: - Error number 1: No memory space available - Error number 2: Maximum possible number of symbols exceeded - Error number 3: Index 1 lies outside the valid value range - Error number 4: Name already exists in channel - Error number 5: Name already exists in NCK If the alarm occurs after cycle programs, macro definitions or definitions for global user data (GUD) have been introduced, the machine data for the user memory configuration has been configured incorrectly. In all other cases, changes to machine data that is already correct lead to errors in the user memory configuration. The following block names (2nd parameter) are known in the NCK (all system and user data blocks; in general, only problems in the user data blocks can be remedied by user intervention): - _N_NC_OPT - System internal: option data, NCK global - _N_NC_SEA - System internal: setting data, NCK global - _N_NC_TEA - System internal: machine data, NCK global - _N_NC_CEC - System internal: 'cross error compensation' - _N_NC_PRO - System internal: protection zones, NCK global - _N_NC_GD1 - User: 1st GUD block defined by _N_SGUD_DEF, NCK global - _N_NC_GD2 - User: 2nd GUD block defined by _N_MGUD_DEF, NCK global - _N_NC_GD3 - User: 3rd GUD block defined by _N_UGUD_DEF, NCK global - _N_NC_GD4 - User: 4th GUD block defined by _N_GUD4_DEF, NCK global - _N_NC_GD5 - User: 5th GUD block defined by _N_GUD5_DEF, NCK global - _N_NC_GD6 - User: 6th GUD block defined by _N_GUD6_DEF, NCK global - _N_NC_GD7 - User: 7th GUD block defined by _N_GUD7_DEF, NCK global - _N_NC_GD8 - User: 8th GUD block defined by _N_GUD8_DEF, NCK global - _N_NC_GD9 - User: 9th GUD block defined by _N_GUD9_DEF, NCK global - _N_NC_MAC - User: macro definitions - _N_NC_FUN - System internal: predefined functions and procedures, NCK global - _N_CHc_OPT - System internal: option data, channel-specific - _N_CHc_SEA - System internal: setting data, channel-specific - _N_CHc_TEA - System internal: machine data, channel-specific - _N_CHc_PRO - System internal: protection zones, channel-specific - _N_CHc_UFR - System internal: frames, channel-specific - _N_CHc_RPA - System internal: R-parameters, channel-specific - _N_CHc_GD1 - User: 1st GUD block defined by _N_SGUD_DEF, channel-specific

- _N_CHc_GD2 - User: 2nd GUD block defined by _N_MGUD_DEF, channel-specific
- _N_CHc_GD3 - User: 3rd GUD block defined by _N_UGUD_DEF, channel-specific
- _N_CHc_GD4 - User: 4th GUD block defined by _N_GUD4_DEF, channel-specific
- _N_CHc_GD5 - User: 5th GUD block defined by _N_GUD5_DEF, channel-specific
- _N_CHc_GD6 - User: 6th GUD block defined by _N_GUD6_DEF, channel-specific
- _N_CHc_GD7 - User: 7th GUD block defined by _N_GUD7_DEF, channel-specific
- _N_CHc_GD8 - User: 8th GUD block defined by _N_GUD8_DEF, channel-specific
- _N_CHc_GD9 - User: 9th GUD block defined by _N_GUD9_DEF, channel-specific
- _N_AXa_OPT - System internal: option data, axial
- _N_AXa_SEA - System internal: setting data, axial
- _N_AXa_TEA - System internal: machine data, axial
- _N_AXa_EEC - System internal: leadscrew error compensation data, axial
- _N_AXa_QEC - System internal: quadrant error compensation data, axial
- _N_TOt_TOC - System internal: toolholder data, TOA-specific
- _N_TOt_TOA - System internal: tool data, TOA-specific
- _N_TOt_TMA - System internal: magazine data, TOA-specific
- _N_NC_KIN - System internal: data to describe kinematic chains, NCK-specific
- _N_NC_NPA - System internal: data to describe 3D protection zones, NCK-specific
- _N_NC_TRA - System internal: transformation data sets, NCK-specific
- _N_NC_WAL - System internal: data to describe coordinate-specific working area limitation
- _N_COMPLETE_CYD - System internal: cycle and display machine data, NCK-, channel-, axis-specific
c = Channel number
a = Machine axis number
t = TOA unit number
There are also other internal system data blocks with identifiers.

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

Remedy:
Correct the machine data or undo the changes made.
Please inform the authorized personnel/service department. There are two determining machine data for cycle programs:
- MD18170 $MN_MM_NUM_MAX_FUNC_NAMES = max. number of all cycle programs, error number = 2 shows that this value is too small.
- MD18180 $MN_MM_NUM_MAX_FUNC_PARAM = max. number of all parameters defined in the cycle programs, error number = 2 shows that this value is too small
(If these MDs are modified, the memory backup is retained)
The following applies to macro definitions:
MD18160 $MN_MM_NUM_USER_MACROS = max. number of all macro definitions, error number = 2 shows that this value is too small.
(If these MDs are modified, the memory backup is retained)
The following applies to GUD variables:
- MD18118 $MN_MM_NUM_GUD_MODULES = max. number of GUD data blocks per area (NCK/channel) (if GD1, GD2, GD3, GD9 are to be defined, then the value must be = 9 and not e.g. = 4).
- MD18120 $MN_MM_NUM_GUD_NAMES_NCK = max. number of all NCK global GUD variables, error number = 2 shows that this value is too small.
- MD18130 $MN_MM_NUM_GUD_NAMES_CHAN = max. number of all channel-specific GUD variables in the channel, error number = 2 shows that this value is too small.
- MD18150 $MN_MM_GUD_VALUES_MEM = total value memory of all GUD variables together, error number = 1 shows that this value is too small.

Program Continuation:
Switch control OFF - ON.

6020   Machine data have been changed - now memory is reorganized

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

Reaction:
Alarm display.

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

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

6030   Limit of user memory has been adapted

Definitions:	Data management checks during power-up the actually available physical user memory (DRAM, DPRAM and SRAM) with the values in the system-specific machine data MD18210 $MN_MM_USER_MEM_DYNAMIC, MD18220 $MN_MM_USER_MEM_DPR und MD18230 $MN_MM_USERMEM_BUFFERED-USERMEM_BUFFERED.
Reaction:	Alarm display.
Remedy:	No remedial measures are required. The new maximum permissible value can be read from the reduced machine data.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Free memory capacity in KB defined for the control model %2 = Actual maximum capacity of free memory in KB %3 = Type of memory, "D" =non-battery-backed, "S" =battery-backed
Definitions:	The alarm can only occur after a 'cold start' (=NCK start-up with standard machine data). The alarm is only a notice. There is no interference with any NCK functions. It shows that the NCK has less free user memory available than specified by Siemens for this control variant. The value of the actually available free user memory can also be taken from the MD18050 $MN_INFO_FREE_MEM_DYNAMIC, MD18060 $MN_INFO_FREE_MEMS_STATIC. Siemens supplies NCK with default settings that, depending on the model, have certain (free) memory space available for the specific settings of the actual applications. The original factory setting of NCK systems is thus that the alarm does not occur with a cold start.
Reaction:	Alarm display.
Remedy:	Reasons for the message: - The NCK contains compile cycle software, that uses so much memory space that the hardware cannot provide the required memory. - The NCK runs on hardware that is not intended for this NCK release (i.e. that has not enough memory capacity). - If the application runs properly with the remaining free user memory (i.e. can be setup without any errors), the message can simply be ignored. - If the actual application cannot be configured because there is not enough memory capacity available, either the existing compile cycle must be reduced or, if possible, the system must be upgraded with additional memory space.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel ID %2 = Block number, label
Definitions:	Occurs when a data management function is called which is not available because ToolMan is deactivated. For example, the language commands GETT, SETPIECE, GETSELT, NEWT, DELT, TCA.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Please inform the authorized personnel/service department. - Make sure of how the NC is supposed to be configured! Is tool management or tool monitoring needed but not activated? - Are you using a part program that is meant for a numerical control with tool management/tool monitoring? It is not possible to start this program on the numerical control without tool management/tool monitoring. Either run the part program on the appropriate NC control or edit the part program. - Activate tool management/tool monitoring by setting the appropriate machine data. See MD18080$MN_MM_TOOL_MANAGEMENT_MASK, MD20310$MC_TOOL_MANAGEMENT_MASK - Check whether the required option is set accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

6436	**[Channel %1:] Block %2 command '%3' cannot be programmed. Function '%4' has not been activated.**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Programmed command %4 = Function identifier
Definitions:	The command cannot be programmed due to the lack of a function enable or activation.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct the NC program
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

6452	**[Channel %1:] Block %2 tool change not possible. Tool holder/spindle number = %3 not defined.**
Parameters:	%1 = Channel ID %2 = Block number, label %3 = Tool holder/spindle number
Definitions:	The desired tool change is not possible. The toolholder/spindle number has not been defined.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	General: The following must apply: 'maximum programmed address extension s (=spindle number/toolholder number) of Ts=t, Ms=6 must be less than the value of MD18076 $MN_MM_NUM_LOCS_WITH_DISTANCE. With magazine management: Check whether the toolholder number/spindle number and the magazine data have been defined correctly. (See also the system variables $TC_MPP1, $TC_MPP5 of the buffer magazine).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

6500	**NC memory full**
Definitions:	The NCK file system is full. The available buffered memory does not suffice. Note: At first commissioning, files of the NC file system may be affected such as drive data, MMC files, FIFO files, NC programs...
Reaction:	Alarm display.
Remedy:	Adjust the size of the buffered memory (MD18230 $MN_MM_USER_MEM_BUFFERED) or increase the space available in the buffered memory, e.g. by unloading part programs that are no longer being used. Or decrease the size of the ring buffer (see $MC_RESU_RING_BUFFER_SIZE).
Program Continuation:	Clear alarm with the Delete key or NC START.

6510	**Too many part programs in the NC memory**
Definitions:	The maximum number of possible files in the NC file system (part of the NC memory) has been reached. Note: During first commissioning, this can affect files from the NC file system, e.g. drive data, MMC files, FIFO files, NC programs, ...
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Delete or unload files (e.g. part programs), or - Increase MD18320 $MN_MM_NUM_FILES_IN_FILESYSTEM.
Program Continuation:	Clear alarm with the Delete key or NC START.

6520	**The value of the machine data %1%2 is too low**
Parameters:	%1 = String: MD identifier %2 = If required, index: MD array
Definitions:	The MD18370 $MN_MM_PROTOC_NUM_FILES specifies the number of protocol files for the protocol users. However, more types are used than configured.
Reaction:	Alarm display.
Remedy:	Increase MD18370 $MN_MM_PROTOC_NUM_FILES.
Program Continuation:	Clear alarm with the Delete key or NC START.

6530	**Too many files in directory**
Definitions:	The number of files in a directory of the NC memory has reached the maximum limit.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Delete or unload files (e.g. part programs) in the respective directory, or - Increase MD18280 $MN_MM_NUM_FILES_PER_DIR.
Program Continuation:	Clear alarm with the Delete key or NC START.

6540	**Too many directories in the NC memory**
Definitions:	The number of directories in the NC file system (part of the NC memory) has reached the maximum limit.
Reaction:	Alarm display.
Remedy:	- Delete or unload directory (e.g. workpiece), or - Increase MD18310 $MN_MM_NUM_DIR_IN_FILESYSTEM.
Program Continuation:	Clear alarm with the Delete key or NC START.

6550	**Too many subdirectories**
Definitions:	The number of subdirectories in a directory of the NCK has reached the maximum limit.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Delete or empty subdirectories in the respective directory, or - Increase MD18270 $MN_MM_NUM_SUBDIR_PER_DIR.
Program Continuation:	Clear alarm with the Delete key or NC START.

6560	**Data format not allowed**
Definitions:	An attempt was made to write impermissible data in an NCK file. This error can occur in particular when the attempt was made to load binary data in the NCK as ASCII file. The error can also occur during preprocessing of cycles (see MD10700 $MN_PREPROCESSING_LEVEL) if the NC block is very long. In this case, subdivide the NC block.
Reaction:	Alarm display.
Remedy:	Specify that the file concerned is a binary file (e.g. extension: .BIN).
Program Continuation:	Clear alarm with the Delete key or NC START.

6570	**NC memory full**
Definitions:	The NC card file system of the NCK is full. The task cannot be executed. Too many system files were created in the DRAM.
Reaction:	Alarm display.
Remedy:	Start fewer "execute from external" processes.

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

6580   NC memory full

Definitions:	The NC card file system of the NCK is full. The task cannot be executed. To many files have been loaded
Reaction:	Alarm display.
Remedy:	Delete or unload files (e.g. parts programs)
Program Continuation:	Clear alarm with the Delete key or NC START.

6581   NC user memory full

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

6582   NC machine OEM memory full

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

6583   NC system memory full

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

6584   NC memory limit TMP reached

Definitions:	The DRAM file system of the TMP (temporary) area is full. The job cannot be executed.
Reaction:	Alarm display.
Remedy:	Increase MD18351 $MN_MM_DRAM_FILE_MEM_SIZE or MD18355 $MN_MM_T_FILE_MEM_SIZE or switch off the precompilation of individual or all cycles or delete files in the TMP area.
Program Continuation:	Clear alarm with the Delete key or NC START.

6585   NC external memory limit reached

Definitions:	The DRAM file system of the external area (execution of the external drive) is full. The job cannot be executed.
Reaction:	Alarm display.
Remedy:	Load the files to be executed explicitly into the NCK.
Program Continuation:	Clear alarm with the Delete key or NC START.

6600   NC card memory is full

Definitions:	The NC card file system of the NCK is full. No more data can be stored on the NC card.
Reaction:	Alarm display.
Remedy:	Delete the data on the PCMCIA card.

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

6610   **Too many files open on NC card**

Definitions:	Too many files are being accessed simultaneously on the NC card.
Reaction:	Alarm display.
Remedy:	Repeat the action later.
Program Continuation:	Clear alarm with the Delete key or NC START.

6620   **NC card has incorrect format**

Definitions:	The NC card cannot be accessed because the format is incorrect.
Reaction:	Alarm display.
Remedy:	Replace the NC card.
Program Continuation:	Clear alarm with the Delete key or NC START.

6630   **NC card hardware is defective**

Definitions:	The NC card cannot be accessed because the card is defective.
Reaction:	Alarm display.
Remedy:	Replace the PCMCIA card.
Program Continuation:	Clear alarm with the Delete key or NC START.

6640   **NC card is not inserted**

Definitions:	The NC card cannot be accessed because the card is not plugged in.
Reaction:	Alarm display.
Remedy:	Plug in the NC card.
Program Continuation:	Clear alarm with the Delete key or NC START.

6650   **Write protection of NC card is active**

Definitions:	The NC card cannot be accessed because the write protection is active.
Reaction:	Alarm display.
Remedy:	Deactivate the write protection.
Program Continuation:	Clear alarm with the Delete key or NC START.

6660   **'Flash File System' option is not set**

Definitions:	The NC card cannot be accessed because the option is not enabled.
Reaction:	Alarm display.
Remedy:	Buy option.
Program Continuation:	Clear alarm with the Delete key or NC START.

6670   **NC card read active**

Definitions:	The alarm is active while the contents of the NC card are being read out. The FFS cannot be accessed during this period.
Reaction:	Alarm display.
Remedy:	Wait until the read-out procedure is terminated.

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

6671　NC card write active

Definitions:	The alarm is active while the contents of the NC card are being written. The flash file system cannot be accessed during this period. If the power is switched off while the alarm is active, the contents of the NC card are destroyed!
Reaction:	Alarm display.
Remedy:	Wait until the write procedure is terminated.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

6690　Cycles from NC card cannot be copied to the passive file system.

Definitions:	There is not enough space in the file system that the directories specified in MD11291 $MN_$DRAM_FILESYST_SAVE_MASK can be copied from the NC card to the passive file system.
Reaction:	Alarm display.
Remedy:	Delete data in the file system.
Program Continuation:	Clear alarm with the Delete key or NC START.

6691　Cycles from the passive file system cannot be saved on the NC card

Definitions:	There is not enough space on the NC card that the directories specified in the $PCMCIA_FUNCTION_MASK can be saved. It is possible that cycles are lost during the next booting.
Reaction:	Alarm display.
Remedy:	Delete data on the NC card or delete cycles not required.
Program Continuation:	Clear alarm with the Delete key or NC START.

6692　Cycle %1 lost

Parameters:	%1 = Name of cycle
Definitions:	A cycle has been changed and due to a power failure, the backup on the PC card could not be terminated properly. The cycle is lost.
Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Import the cycle again.
Program Continuation:	Switch control OFF - ON.

6693　File %1 lost

Parameters:	%1 = File name
Definitions:	Due to a power failure, a file change could not be terminated properly. The file is lost.
Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Import the file again.
Program Continuation:	Switch control OFF - ON.

6698	Unknown NC card (%1/%2). Writing not possible.
Parameters:	%1 = actManufactorCode (manufacturer code read by the card) %2 = actDeviceCode (memory code read by the card)
Definitions:	The NC card cannot be accessed because a valid write algorithm is not available for the flash memory.
Reaction:	Alarm display.
Remedy:	Use a compatible NC card or enter the new manufacturer code/device code in MD11700 $MN_PERMISSIVE_FLASH_TAB after consultation with SIEMENS.
Program Continuation:	Clear alarm with the Delete key or NC START.

6700	[Channel %1:] Value of the machine data %2%3 is too low
Parameters:	%1 = Channel number %2 = MD identifier %3 = If required, field index
Definitions:	The MD28302 $MC_MM_PROTOC_NUM_ETP_STD_TYP specifies the number of default event types for the protocol users. However, more types are used than configured.
Reaction:	Alarm display.
Remedy:	Increase MD28302 $MC_MM_PROTOC_NUM_ETP_STD_TYP.
Program Continuation:	Clear alarm with the Delete key or NC START.

7500	Block %1 invalid protection level for command %2 (protection level act.: %3 prog.: %4)
Parameters:	%1 = Block number %2 = Programmed command %3 = Current protection level of the command %4 = Programmed protection level of the command
Definitions:	On assigning a protection level for a parts program command via REDEF command - an impermissible parts program command has been programmed - a protection level has been programmed that is logically smaller (larger in value) than the protection level currently applicable for this command. - the relevant definition file has not been protected sufficently against write access. The write protection of the file must be at least as high as the highest protection level that has been assigned to a parts program command in this definition file.
Reaction:	Alarm display.
Remedy:	Modify definition files /_N_DEF_DIR/_N_MACCESS_DEF or /_N_DEF_DIR/_N_UACCESS_DEF-CESS_ DEF. Please see the Siemens Programming Guide or the OEM documentation for the language commands permissible for the relevant system configurations.
Program Continuation:	Clear alarm with the RESET key. Restart part program

8000	[Channel %1:] Option 'Interrupt routines' not set
Parameters:	%1 = Channel number
Definitions:	Fast NCK inputs are required for the input signals in order to activate the interrupt routines and rapid lift from contour. This function is not included in the basic version and must be retrofitted when needed.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Do not use rapid interrupt inputs or contact the machine manufacturer with a view to retrofitting this option!
Program Continuation:	Clear alarm with the RESET key. Restart part program

8010	**Option 'activation of more than %1 axes' not set**
Parameters:	%1 = Number of axes
Definitions:	More machine axes have been defined through the MD20070 $MC_AXCONF_MACHAX_USED than are allowed in the system.
Reaction:	NC not ready. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. The sum of all axes that have been configured using the channel-specific MD20070 $MC_AXCONF_MACHAX_USED must not exceed the maximum number of axes (dependent upon the configuration -> option, basic version: 4 axes). Please also observe the information relating to the definition of auxiliary axes/spindles.
Program Continuation:	Switch control OFF - ON.

8022	**Option 'activation of more than %1KB SRAM' not set**
Parameters:	%1 = Memory size
Definitions:	The option for memory extension does not correspond to the active SRAM.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Buy option - Activate less SRAM
Program Continuation:	Switch control OFF - ON.

8025	**[Channel %1:] Option 'Advanced Surface' not set**
Parameters:	%1 = Channel number
Definitions:	The option for 'Advanced Surface' functionality is not set.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Purchase option - Reset the activation of 'Advanced Surface' functionality (MD20606 $MC_PREPDYN_SMOOTHING_ON and/or MD20443 $MC_LOOKAH_FFORM)
Program Continuation:	Switch control OFF - ON.

8030	**[Channel %1:] Block %2 option 'interpolation of more than %3 axes' not set**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of permissible axes
Definitions:	The option for the number of interpolating axes does not correspond to the number of axes programmed in the interpolation group.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Option: "Interpolation of more than 4 axes" (the number of axes permitted can be set in this option) or, in the part program, program up to as many axes as are permitted by the configuration of the controller.

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

---

8031   [Channel %1:] Block %2 axis %3: Axis has no IPO functionality

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis, spindle number
Definitions:	An axis/spindle that has been defined as a special axis/auxiliary spindle (see MD30460 $MA_BASE_FUNCTION_MASK bit8), should be operated as an interpolating axis.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Define axis as interpolating axis (see MD30460 $MA_BASE_FUNCTION_MASK bit8) or change part program
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

8037   'Activate APC/Number of current setpoint filters' option not set.

Definitions:	More than six current setpoint filters were activated in the drive, although the corresponding option had not been set.
Reaction:	NC not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	- Buy option - Deactivate the 'Advanced Positioning Control' (APC) function in the drive. - Set a maximum of six current setpoint filters in the drive.
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = String: MD identifier
Definitions:	A machine data has been set that is locked by an option.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. For retrofitting the option, please refer to your machine manufacturer or to a sales representative of SIEMENS AG, A&D MC.
Program Continuation:	Clear alarm with the Delete key or NC START.

8041   Axis %1: MD %2 reset, corresponding option not sufficient

Parameters:	%1 = Axis number %2 = String: MD identifier
Definitions:	All of the axes selected in the machine data of the assigned option are used. Safety functions have been selected for too many axes in the axial machine data. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	--

Program Continuation:	Switch control OFF - ON.

8044   Option for IPO cycle time %1 ms not set

Parameters:	%1 = Impermissible IPO cycle time
Definitions:	The option for activation of an IPO cycle time of %1 ms has not been set. Option - Permiss. IPO cycle time: - Option-free >= 8ms - 1. 1st step >= 6ms - 2. 2nd step >= 4ms - 3. 3rd step >= 2ms - 4. 4th step <2ms
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	- Buy option - Increase IPO cycle time (e.g. via MD10070 $MN_IPO_SYSCLOCK_TIME_RATIO)
Program Continuation:	Switch control OFF - ON.

8051   Option 'Handwheel on PROFIBUS' not set

Definitions:	The option to operate handwheels on PROFIBUS is not set.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Activate option 'Handwheel on PROFIBUS'
Program Continuation:	Switch control OFF - ON.

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

Parameters:	%1 = Number of non-licensed options
Definitions:	One or more options were activated but no license key was set to prove the purchase of the option(s).
Reaction:	Alarm display.
Remedy:	Generate a license key on the internet at http://www.siemens.com/automation/licence and enter it in the operating area "Setup", function (HSK) "Licenses".
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Parameters:	%1 = Number of non-licensed options
Definitions:	One ore more options were activated, that are not licensed by the license key entered.
Reaction:	Alarm display.
Remedy:	Generate a new license key on the internet at http://www.siemens.com/automation/licence and enter it in the operating area "Setup", function (HSK) "Licenses"..
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Definitions:	The license key was entered wrongly at least three times. Before the next input, a new power ON is required.
Reaction:	Alarm display.
Remedy:	Execute NCK Power On and enter the license key (correctly).
Program Continuation:	Clear alarm with the Delete key or NC START.

8083	Export-restricted system software without valid licensing.
Definitions:	A special CompactFlash Card and a special license key are required for operating the export-restricted system software.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please ensure that a corresponding CompactFlash Card is available in the controller. Generate a license key for export-restricted system software via the internet under http://www.siemens.com/automation/licence and enter in the license key under "Start-up" operating area function (HSK) "Licenses".
Program Continuation:	Switch control OFF - ON.

8100	[Channel %1:] Block %2: function not possible
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	- Impossible due to embargo regulations: - 1. Synchronous actions: Writing of feed, override and axial offsets ($AA_VC, $AC_VC, $AA_OVR, $AA_VC and $AA_OFF) from synchronous actions as well as Continuous Dressing can be programmed only once in a block. - 2. Extended measurement: 'Cyclic measurement' (MEAC) and 'Measurement from synchronous action' is not possible. - 3. Axis interpolation: The number of axes interpolating with one another must not exceed 4 (this also includes synchronous coupling of axes via synchronous actions "DO POS[X]=$A..." "DO FA[X]=$A...").
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

8102	[Channel %1:] Block %2 motion synchronous action: %3 function not possible
Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	- Impossible due to embargo regulations: - 1. Synchronous actions: Writing of feed, override and axial offsets ($AA_VC, $AC_VC, $AA_OVR, $AA_VC and $AA_OFF) from synchronous actions as well as Continuous Dressing can be programmed only once in a block. - 2. Extended measurement: 'Cyclic measurement' (MEAC) and 'Measurement from synchronous action' is not possible. - 3. Axis interpolation: The number of axes interpolating with one another must not exceed 4 (this also includes synchronous coupling of axes via synchronous actions "DO POS[X]=$A..." "DO FA[X]=$A...").
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

9000	Handwheel %1 failed
Parameters:	%1 = Handwheel number
Definitions:	
Reaction:	Interface signals are set. Alarm display.

Remedy:	Restore connection to PROFIBUS handwheel
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	NC start has been activated in the MDI or AUTOMATIC mode and at least one axis that needs to be referenced has not reached its reference point.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Via the channel-specific MD20700: $MC_REFP_NC_START_LOCK (NC Start without reference point) you can decide whether or not the axis has to be referenced before NC Start. The start of referencing can be enabled channel-specific or axis-specific. Channel-specific reference point approach: The rising edge of the NC/PLC interface signal DB3200 DBX1.0 (Activate referencing) starts an automatic sequence which starts the axes of the channel in the same sequence as specified in the axis-specific MD34110 $MA_REFP_CYCLE_NR (axis sequence channel-specific referencing). 0: The axis does not participate in channel-specific referencing, but it must be referenced for NC Start, -1: The axis does not participate in channel-specific referencing, but it need not be referenced for NC Start, 1- 8: Starting sequence for the channel-specific referencing (simultaneous start at the same no.), 1 - 31: CPU type Axis-specific referencing: Press the direction key that corresponds to the approach direction in the axis-specific MD34010 $MA_REFP_CAM_MDIR_IS_MINUS (reference point approach in minus direction).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10208 [Channel %1:] Continue program with NC start

Parameters:	%1 = Channel number
Definitions:	After block search with calculation, the control is in the desired state. The program can now be started with NC Start or the state can be changed for the time being with overstore/jog.
Reaction:	Interpreter stop Alarm display. NC Stop on alarm.
Remedy:	Press NC Start.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10209 [Channel %1:] Internal NC stop after block search

Parameters:	%1 = Channel number
Definitions:	Internal alarm which only initiates the alarm response NC Stop. The alarm is output in the following situations: - If MD11450 $MN_SEARCH_RUN_MODE, bit 0 ==1 and the last action block is loaded in the main run after block search. Alarm 10208 is then activated as a function of the NC/PLC interface signal DB3200 DBX1.6 (PLC action finished). - Search alarm 10208 has been suppressed by the PI service _N_FINDBL (third decade of the parameter supplied with "2"). Alarm 10209 is set as a function of whether or not a search ASUB has been configured (MD11450 $MN_SEARCH_RUN_MODE bit 1) with the end of the search ASUB or the loading of the last action block in the main run.
Reaction:	Interpreter stop NC Stop on alarm.
Remedy:	NC-Start
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10225 [Channel %1:] command denied

Parameters:	%1 = Channel number

Definitions:	The channel has received a command that cannot be executed.
Reaction:	Alarm display.
Remedy:	Press RESET.
Program Continuation:	Clear alarm with the Delete key or NC START.

10226 [Channel %1:] Reset/program end aborted

Parameters:	%1 = Channel number
Definitions:	An error occurred during reset or program end, so that the channel cannot be switched to a ready state. This can occur, for example, if the interpreter reports an error during the processing of the init. blocks created during reset and program end. As a rule, further alarms indicate the exact problem.
Reaction:	NC Start disable in this channel. Channel not ready. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Rectify the problem indicated by the other alarms, and press RESET again.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel number
Definitions:	The Auto-Repos function (operating mode) was selected in the channel but is not implemented.
Reaction:	Alarm display.
Remedy:	This message is purely informational.
Program Continuation:	Clear alarm with the Delete key or NC START.

10600 [Channel %1:] Block %2 auxiliary function during thread cutting active

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An auxiliary function output is programmed in a thread cutting block.
Reaction:	Alarm display.
Remedy:	Consequential errors can occur if the machining path of the thread block is too short and further blocks (thread blocks) follow in which no machining stop may occur. Possible remedial measures: - Program a longer path and/or a lower traversing rate. - Output auxiliary function in another block (program section).
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm occurs only when several blocks with G33 follow in succession. The block end velocity in the specified block is zero, although a further thread cutting block follows. The reasons for this can be, for instance: - G9 - Auxiliary function after motion - Auxiliary function output before the motion of the following block - Positioning axis in the block
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Modify the NC part program by removing any programmed "Stop at end of block" G09.
	Modify general MD11110 $MN_AUXFU_GROUP_SPEC [n] for selecting the output time of an auxiliary function group by changing "Auxiliary function output before/after the movement" to "Auxiliary function output during the movement".
	Bit 5 = 1: Auxiliary function output before movement
	Bit 6 = 1: Auxiliary function output during movement
	Bit 7 = 1: Auxiliary function output after movement
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The thread pitch increase is causing an axis overload. A spindle override of 100% is assumed during verification.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
Remedy:	Reduce the spindle speed, thread pitch increase or path length in the NC program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The thread pitch decrease is causing an axis standstill in the thread block.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
Remedy:	Reduce the thread pitch decrease or path length in the NC program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	An axis/spindle has been positioned over several NC blocks using the POSA/SPOSA instruction. The programmed target position had not yet been reached ("exact stop fine" window) when the axis/spindle was reprogrammed.
	Example:
	N100 POSA[U]=100
	:
	N125 X... Y... U... ; e.g.: U axis still travels from N100!
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Check and correct the part program (analyze whether motion beyond block boundaries is appropriate here). Prevent block change by means of the keyword WAITP for axes or WAITS for spindles until the positioning axes or positioning spindles have also reached their target position.
	Example for axes:
	N100 POSA[U]=100
	:
	N125 WAITP(U)
	N130 X... Y... U...
	Example for spindles:
	N100 SPOSA[2]=77
	:

N125 WAITS(2)
N130 M6

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

---

10620 [Channel %1:] Block %3 axis %2 at software limit switch %4

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label %4 = String
Definitions:	During the traversing motion, the system detected that the software limit switch would be traversed in the direction indicated. Exceeding the traversing range was not detected during block preparation because there has either been a motion overlay or a work offset has been executed or a coordinate transformation is active.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Depending on the reason for this alarm being triggered, the following remedial measures should be undertaken: - Handwheel override: Cancel the motion overlay and avoid this or keep it smaller when the program is repeated. - Transformation: Check the preset/programmed work offsets (current frame). If the values are correct, the tool holder (fixture) must be moved in order to avoid triggering the same alarm when the program is repeated, which would again cause the program to be aborted.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = String %4 = The axis of the software limit switch is only output if different from the traversing axis.
Definitions:	The specified axis is already positioned at the displayed software end delimiter.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check machine data MD36110 $MA_POS_LIMIT_PLUS/ MD36130 $MA_POS_LIMIT_PLUS2 and MD36100 $MA_POS_LIMIT_MINUS/MD36120 $MA_POS_LIMIT_MINUS2 for the software limit switches. Shut down in JOG mode from the software limit switch. Please inform the authorized personnel/service department. Machine data: Check whether the 2nd software limit switch has been selected in the.axis-specific interface signals: "DB380x DBX1000.3 (2nd software limit switch plus) and DB380x DBX1000.2 (2nd software limit switch minus).
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10630 [Channel %1:] Block %2 axis %3 at working area limit %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis, spindle number %4 = String (+ or -)
Definitions:	The specified axis violates the working area limitation. This is recognized only in the main run either because the minimum axis values could not be measured before the transformation or because there is a motion overlay.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Program other motion or do not perform overlaid motion.

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

---

10631	**[Channel %1:] Axis %2 rests at working area limit %3%4**
Parameters:	%1 = Channel number %2 = Axis, spindle %3 = String (+ or -) %4 = The axis of the working area limitation is only output if different from the traversing axis.
Definitions:	The specified axis reaches the working area limitation in JOG mode.
Reaction:	Alarm display.
Remedy:	Check SD43420 $SA_WORKAREA_LIMIT_PLUS and SD43430 $SA_WORKAREA_LIMIT_MINUS for the working area limitation.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10700	**[Channel %1:] Block %2 NCK protection zone %3 violated during automatic or MDI mode**
Parameters:	%1 = Channel number %2 = Block number %3 = Protection zone number
Definitions:	The workpiece-related NCK protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Protection zone can be traversed after a new NC Start.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10701	**[Channel %1:] Block %2 channel-specific protection zone %3 violated during automatic or MDI mode**
Parameters:	%1 = Channel number %2 = Block number %3 = Protection zone number
Definitions:	The workpiece-related channel-specific protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Protection zone can be traversed after a new NC Start.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10702	**[Channel %1:] NCK protection zone %2 violated during manual mode**
Parameters:	%1 = Channel number %2 = Protection zone number
Definitions:	The workpiece-related NCK protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Protection zone can be traversed after a new NC Start.

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

10703 [Channel %1:] Channel-specific protection zone %2 violated during manual mode

Parameters:	%1 = Channel number %2 = Protection zone number
Definitions:	The workpiece-related channel-specific protection zone has been violated. Note that another tool-related protection zone is still active. The workpiece-related protected area can be traversed after a new NC Start.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Protection zone can be traversed after a new NC Start.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10704 [Channel %1:] Block %2 protection zone monitoring is not guaranteed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	New movements of a geometry axis which have been added could not be allowed for at the time of block preparation. It is therefore not certain that the protection zones will not be violated. This is just a warning message without further reactions.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Take other measures to ensure that the geomotry axes motion, including the additional motion, does not violate the protection zones. (The warning comes nevertheless) or exclude additional motions.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10706 [Channel %1:] NCK protection zone %2 reached with axis %3 during manual mode

Parameters:	%1 = Channel number %2 = Protection zone number %3 = Axis name
Definitions:	The workpiece-related NCK protection zone has been reached with the specified axis. Note that another tool-related protection zone is still active. The workpiece-related protection zone can be traversed when the PLC has issued an enable signal.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Protection zone can be traversed after enable signal from PLC.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10707 [Channel %1:] Channel-specific protection zone %2 reached with axis %3 during manual mode

Parameters:	%1 = Channel number %2 = Protection zone number %3 = Axis name
Definitions:	The workpiece-related channel-specific protection zone has been reached with the specified axis. Note that another tool-related protection zone is still active. The workpiece-related protection zone can be traversed when the PLC has issued an enable signal.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. Protection zone can be traversed after enable signal from PLC.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

10720   [Channel %1:] Block %3 axis %2 software limit switch %4

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label %4 = String (+ or -)
Definitions:	The path programmed for the axis violates the currently valid software limit switch. The alarm is activated when preparing the part program block. If bit 11=0 in the machine data MD11411 $MN_ENABLE_ALARM_MASK, this alarm is issued instead of alarm 10722. If bit 11 is set in the machine data MD11411 $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Check and correct positions in the NC program. Please inform the authorized personnel/service department. Check machine data: MD36100 $MA_POS_LIMIT_MINUS / MD36120 $MA_POS_LIMIT_MINUS2 and MD36110 $MA_POS_LIMIT_PLUS / MD36130 $MA_POS_LIMIT_PLUS2 for the software limit switches. Check the axis-specific interface signals: DB380x DBX1000.3 / .2 (2nd software limit switch plus/minus) to see whether the 2nd software limit switch is selected. Check the currently active work offsets via the current frame. Work offsets, overlaid movements ($AA_OFF), DRF and transformation components must also be checked.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10721   [Channel %1:] Block %3 axis %2 software limit switch %4

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label %4 = String (+ or -)
Definitions:	The motion planned for the axis violates the currently valid software limit switch. The alarm is activated during the preprocessing of approach or rest blocks for REPOS. Depending on MD11411 $MN_ENABLE_ALARM_MASK, bit11=0 this alarm is output instead of alarm 10723. If this MD11411 $MN_ENABLE_ALARM_MASK, bit11 is set in this machine data $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI. See also the Diagnostics Manual for alarm 10723.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs. Check the axis-specific NC/PLC interface signals DB380x DBX1000.3 / .2 (2nd software limit switch plus/minus) to see whether the 2nd software limit switch is selected. Check the currently active work offset via the current frame. Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components. Cancel the NC program with NC reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|number+string(+/-)|residual distance
Definitions:	The path programmed for the axis violates the currently valid software limit switch. The alarm is activated when preparing the part program block. This alarm is issued instead of alarm 10720 if bit 11=1 in the MD11411 $MN_ENABLE_ALARM_MASK. Alarm 10722 offers an expanded diagnostics option for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI. See also diagnostics guide for alarm 10720.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Check and correct positions in the NC program. Please inform the authorized personnel/service department. Machine data: MD36100 $MA_POS_LIMIT_MINUS/MD36120 $MA_POS_LIMIT_MINUS2 and MD36110 $MA_POS_LIMIT_PLUS/MD36130 $MA_POS_LIMIT_PLUS2 must be checked for the software limit switches. Check the axis-specific interface signals: DB380x DBX1000.3 / .2 (2nd software limit switch plus/minus) to see whether the 2nd software limit switch is selected. Check currently active work offsets via the current frame. Work offsets, overlaid movements ($AA_OFF), DRF and transformation components must also be checked.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|number+string(+/-)|residual distance
Definitions:	The motion planned for the axis violates the currently active software limit switch. The alarm is activated during the preprocessing of approach or rest blocks for REPOS. This alarm is issued instead of alarm 10721 if bit11=1 in MD11411 $MN_ENABLE_ALARM_MASK. Alarm 10723 offers an expanded diagnostics option for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI. See also the Diagnostics Manual for alarm 10721.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs. Please inform the authorized personnel/service department. Check the MD36100 $MA_POS_LIMIT_MINUS / MD36120 $MA_POS_LIMIT_MINUS2 and MD36110 $MA_POS_LIMIT_PLUS / MD36130 $MA_POS_LIMIT_PLUS2 for the software limit switches. Check the axis-specific interface signals DB380x DBX1000.3 / .2 (2nd software limit switch plus/minus) to see whether the 2nd software limit switch is selected. Check the currently active work offset via the current frame. Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components. Cancel the NC program with NC reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10730 [Channel %1:] Block %3 axis %2 working area limitation %4

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label %4 = String (+ or -)
Definitions:	This alarm is generated if it is determined during block preparation that the programmed path of the axis violates the working area limitation. If bit 11=0 in machine data MD11411$MN_ENABLE_ALARM_MASK, this alarm is issued instead of alarm 10732. If bit 11 is set in machine dataMD11411 $MN_ENABLE_ALARM_MASK, an expanded diagnostics option is offered for the software limit switch violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	a) Check NC program for correct positional data and, if necessary, make corrections. b) Check work offsets (current frame) c) Correct working area limitation via G25/G26, or d) Correct working area limitation via setting data, or e) Deactivate working area limitation via setting data 43410 WORKAREA_MINUS_ENABLE=FALSE
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10731 [Channel %1:] Block %3 axis %2 working area limitation %4

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label %4 = String (+ or -)
Definitions:	The motion planned for the axis violates the currently active working area limit. The alarm is activated during the preparation of approach or rest blocks for REPOS. This alarm is issued instead of alarm10733 if bit11 is not set in MD11411 $MN_ENABLE_ALARM_MASK.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs. Check the currently active work offset via the current frame. Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components. Cancel NC program with NC reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|residual distance
Definitions:	This alarm is generated if it is determined during block preprocessing that the programmed path of the stated axis violates the working area limitation. If MD11411 $MN_ENABLE_ALARM_MASK, bit11=1, this alarm is issued instead of alarm 10730. Alarm 10732 offers an expanded diagnostics option for the working area limitation violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.

Remedy:	a) Check NC program for correct positional data and, if necessary, make corrections. b) Check work offsets (current frame) c) Correct working area limitation via G25/G26, or d) Correct working area limitation via setting data, or e) Deactivate working area limitation via SD43410 $SA_WORKAREA_MINUS_ENABLE=FALSE
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|residual distance
Definitions:	The motion planned for the axis violates the currently active working area limitation. The alarm is activated during the preparation of approach or rest blocks for REPOS. This alarm is issued instead of alarm 10731 if bit11=1 in MD11411 $MN_ENABLE_ALARM_MASK. Alarm 10733 offers an expanded diagnostics option for the working area limitation violation. The condition for activation is the presence of the ALUN* alarm file in the HMI.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs. Check the currently active work offset via the current frame. Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components. Cancel NC program with NC reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|residual distance
Definitions:	This alarm is generated if it is determined during block preparation that the programmed path of the stated axis violates the coordinate system-specific working area limitation.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	a) Check NC program for correct positional data and, if necessary, make corrections. b) Check work offsets (current frame) c) Correct the working area limitation with WALCS1 ... WALCS9, or d) Correct the working area limitation in $P_WORKAREA_CS_LIMIT_PLUS or $P_WORKAREA_CS_LIMIT_MINUS, or e) Deactivate the working area limitation with $P_WORKAREA_CS_MINUS_ENABLE =FALSE or $P_WORKAREA_CS_PLUS_ENABLE. In cases d) and e), then reactivate the group of the selected coordinate system-specific working area limitation.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10736	**[Channel %1:] Block %5 axis %2 coordinate system-specific working area limitation violated, residual distance: %6 %3<ALUN>**
Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Unit of distance %4 = Block number, label|residual distance
Definitions:	This alarm is generated if it is determined during block preparation that the programmed path of the stated axis violates the coordinate system-specific working area limitation. The alarm is activated during the preparation of approach or residual blocks for REPOS.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Determine the cause of the offset from the initial or target position. The REPOS command is executed at the end of an ASUB or system ASUB. See also cross reference from ASUBs. Check the currently active work offset via the current frame. Also check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components. Cancel NC program with NC reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10740	**[Channel %1:] Block %2 too many empty blocks in WAB programming**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	It is not allowed to program more blocks than specified by MD20202 $MC_WAB_MAXNUM_DUMMY_BLOCKS between the WAB block and the block determining the approach and retraction tangent.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10741	**[Channel %1:] Block %2 direction reversal with WAB infeed motion**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A safety distance which has been programmed is located perpendicular to the machining plane and not between the start and end point of the WAB contour.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10742	**[Channel %1:] Block %2 WAB distance invalid or not programmed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Possible causes: In a WAB block, the parameter DISR has not been specified or its value is less than or equal to 0.

During approach or retraction with circle and active tool radius, the radius of the internally generated - WAB contour is negative. The internally generated WAB contour is a circle with a radius which, when offset with the current offset radius (sum of tool radius and offset value OFFN), yields the tool center point path with the programmed radius DISR.

Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10743 [Channel %1:] Block %2 WAB programmed several times

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt has been made to activate a WAB motion before a previously activated WAB motion was terminated.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10744 [Channel %1:] Block %2 no valid WAB direction defined

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The tangent direction for smooth approach or retraction is not defined. Possible causes: In the program, no block with travel information follows the approach block. Before a retraction block, no block with travel information has been programmed in a program. The tangent to be used for WAB motion is vertical to the current machining plane.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the WAB block and in the following block, the position has been programmed perpendicular to the machining direction. In the WAB block, no position has been indicated in the machining plane.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program. Either remove the position data for the infeed axis from the WAB block or the following block, or program a position in the machining plane in the WAB block as well.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A preprocessing stop has been inserted between an SAR approach block and the following block defining the tangent direction or between an SAR retraction block and the following block defining the end position.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10747   [Channel %1:] Block %2 retraction direction not defined for WAB

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a WAB retraction block with quarter circle or semi-circle (G248 or G348), the end point in the machining plane was not programmed, and either G143 or G140 without tool radius compensation is active.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program. The following changes are possible: - Indicate end point in the machining plane in the WAB block. - Activate tool radius compensation (effective for G140 only, not for G143). - State retraction side explicitly with G141 or G142. - Perform retraction with a straight line instead of a circle.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	By means of DISRP a position of the retraction plane has been programmed which is not situated between the safety distance (DISCL) and the starting point (during approach) and/or end point (during retraction) of the WAB movement.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10750   [Channel %1:] Block %2 tool radius compensation activated without tool number

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A tool T... must be selected so that the control can make allowance for the associated compensation values. A correction data block (D1) containing the correction values (parameter P1 - P25) is automatically assigned to each tool (T number). Up to 9 correction data blocks can be assigned to a tool by specifying the required data block with the D number (D1 - D9). The cutter radius compensation (CRC) is allowed for if function G41 or G42 is programmed. The correction values are contained in parameter P6 (geometry value) and P15 (wear value) of the active correction data block Dx.

Reaction:	Correction block is reorganized. Interpreter stop Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Before calling the CRC with G41/G42, program a tool number under the address T...
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The "Bottleneck detection" (calculation of intersection for the following compensated traversing blocks) has not been able to calculate a point of intersection for the reviewed number of traversing blocks. It is therefore possible that one of the equidistant paths violates the workpiece contour.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Please inform the authorized personnel/service department. Check the part program and, if possible, modify the programming so that inside corners with smaller paths than the correction value are avoided. (Outside corners are not critical because the equidistants are lengthened or intermediate blocks are inserted, so that there is always a point of intersection). Increase the number of reviewed traversing blocks via machine data MD20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS (default: 3), this increases the amount of calculation and consequently also the block cycle time.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The cutter radius compensation must buffer a variable number of intermediate blocks in order to enable calculation of the equidistant tool path for each NC block. The size of the buffer cannot be determined by simple means. It depends on the number of blocks without traversing information in the compensation plane, the number of contour elements to be inserted and the shape of the curvature in spline and polynomial interpolation. The size of the buffer is fixed by the system and cannot be changed via the MDs.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Please inform the authorized personnel/service department. Reduce the size of the buffer that has been assigned by modifying the NC program. - By avoiding: - Blocks without traversing information in the compensation plane - Blocks with contour elements having a variable curvature (e.g. ellipses) and with curvature radii that are smaller than the compensation radius. (Such blocks are divided up into several subblocks). - Reduce the number of reviewed blocks for collision monitoring (MD20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

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

Definitions:	Selection of tool radius compensation with G41/G42 may only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active.
	In the block with G41/G42, at least one axis in the plane G17 to G19 must be written. It is always advisable to write both axes because, as a rule, both axes are traversed when selecting the compensation.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Correct the NC program and put the compensation selection in a block with linear interpolation.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10754   [Channel %1:] Block %2 deselection of the tool radius compensation only possible in linear block

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Deselection of tool radius compensation with G40 can only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active.
	In the block with G40, at least one axis in the plane G17 to G19 must be written. It is always advisable to write both axes because, as a rule, both axes are traversed when deselecting the compensation.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Correct the NC program and put the compensation selection in a block with linear interpolation.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10755   [Channel %1:] Block %2 selection of the tool radius compensation via KONT not possible at the current starting point

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	When activating the cutter radius compensation with KONT the starting point of the approach block is within the compensation circle and therefore already violates the contour.
	If the cutter radius compensation is selected with G41/G42, the approach behavior (NORM or KONT) determines the compensation movement if the present actual position is behind the contour. With KONT, a circle is drawn with the cutter radius around the programmed initial point (= end point of the approach block). The tangent that passes through the current actual position and does not violate the contour is the approach movement.
	If the start point is within the compensation circle around the target point, no tangent passes through this point.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Place selection of the CRC such that the starting point of the approach movements comes to rest outside of the correction circle around the target point (programmed traversing movements > compensation radius). The following possibilities are available:
	Selection in the previous block
	Insert intermediate block
	Select approach behavior NORM
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10756 [Channel %1:] Block %2 deselection of the tool radius compensation via KONT not possible at the programmed end point

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	On deselection of the cutter radius compensation, the programmed end point is within the compensation circle. If this point were in fact to be approached without compensation, there would be a contour violation.
	If the cutter radius compensation is deselected via G40, the approach behavior (NORM or KONT) determines the compensation movement if the programmed end point is behind the contour. With KONT, a circle is drawn with the cutter radius about the last point at which the compensation is still active. The tangent passing through the programmed end position and not violating the contour is the retraction movement.
	If the start point is within the compensation circle around the target point, no tangent passes through this point.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Place deselection of the CRC such that the programmed end point comes to rest outside the compensation circle around the last active compensation point. The following possibilities are available:
	Deselection in the next block
	Insert intermediate block
	Select retract behavior NORM
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	In order to change the compensation plane (G17, G18 or G19) it is first necessary to deselect the cutter radius compensation with G40.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Insert an intermediate block in the part program using the correction deselection. After the plane change, the cutter radius compensation is to be selected in an approach block with linear interpolation.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10758 [Channel %1:] Block %2 curvature radius with variable compensation value too small

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The current cutter radius compensation (the cutter used) is too large for the programmed path radius.
	In a block with variable tool radius compensation, a compensation must be possible either anywhere or nowhere on the contour with the smallest and the largest compensation value from the programmed range. There must be no point on the contour in which the curvature radius is within the variable compensation range.
	If the compensation value varies its sign within a block, both sides of the contour are checked, otherwise only the compensation side.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Use smaller cutters or allow for a part of the cutter radius at the time of contour programming.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10759 [Channel %1:] Block %2 path is parallel to tool orientation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a block with spline or polynomial interpolation, the corrected path runs in at least one point parallel to the tool orientation, i.e. the path has a tangent perpendicular to the compensation plane. The tangent at a point on a path is regarded as parallel to the tool orientation if the angle between the two directions is less than the limit value defined by MD21080 $MC_CUTCOM_PARALLEL_ORI_LIMIT. However, in circumferential milling, straight lines running parallel to the tool orientation are permissible, as well as circles with a circle plane perpendicular to the compensation plane (application with smooth retraction from the groove). Straight lines in the direction of the tool orientation are not permissible in face milling (CUT3D, CUT3DF, CUT3DFS).
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Do not use splines or polynomials when writing the contour section, but straight lines and circles instead. Divide up the tool piece geometry and deselect the cutter radius compensation between the various sections.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10760 [Channel %1:] Block %2 helical axis is not parallel to tool orientation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	With active tool radius compensation a helix is only permissible if the helix axis is parallel to the tool, i.e. the circle plane and the compensation plane must be identical.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Orient helix axis perpendicular to the machining plane.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10761 [Channel %1:] Block %2 tool radius compensation for ellipse with more than one revolution not possible

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When machining the inside of an ellipse, in parts of the ellipse the curvature radii are greater than or smaller than the cutter radius compensation. In ellipses, in this case the block would be split up into 4 subblocks with curvature radii that are greater than and less than the compensation radius. Over several revolutions, there would be a tremendous increase in the amount of calculation required by the unlimited number of resulting subblocks, and therefore this situation is rejected by the error message. If compensation is possible everywhere or nowhere on the ellipse, then ellipses are also permissible that cover more than one full revolution.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Use cutter with smaller radius or program motion block on blocks with no more than one revolution.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10762   **[Channel %1:] Block %2 too many empty blocks between two traversing blocks with active tool radius compensation**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The maximum permissible number of empty blocks is limited by a machine data.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	- Modify part program - Modify machine data - Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Due to collision monitoring with active tool radius compensation, the path component of the block in the compensation plane becomes zero. If the original block contains no motion information perpendicular to the compensation plane, it means that this block is excluded. The alarm can be suppressed with MD11410 $MN_SUPPRESS_ALARM_MASK bit1 = 1.
Reaction:	Alarm display.
Remedy:	- The behavior is correct at narrow locations that cannot be machined with the active tool. - Modify the part program if necessary. - Use tool with smaller radius if necessary. - Program CDOF/CDOF2.
Program Continuation:	Clear alarm with the Delete key or NC START.

10764   **[Channel %1:] Block %2 discontinuous path with active tool radius compensation**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm occurs when, with active tool radius compensation, the starting point used for calculating the compensation is not identical to the end point of the preceding block. This situation can occur, for example, when a geometry axis is traversed between two positions as a positioning axis or when, with an active kinematic transformation (e.g. 5-axis transformation) the tool length compensation is altered.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10765   **[Channel %1:] Block %2 3D tool radius compensation not possible**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm occurs when an attempt is made to activate the 3D tool radius compensation even though the option required for this is not fitted in the control.
Reaction:	Correction block is reorganized.

Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

| **Remedy:** | The option cannot be activated by altering machine data because the necessary code is not physically available. |
| **Program Continuation:** | Clear alarm with NC START or RESET key and continue the program. |

10770 [Channel %1:] Block %2 change of corner type due to change of orientation with active tool radius compensation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The type of a corner (inside or outside corner) depends not only on the programmed path but also on the tool orientation. For this purpose, the programmed path is projected in the plane perpendicularly to the actual tool orientation and the corner type is determined there. If a change in orientation is programmed (in one or several blocks) between two traversing blocks, resulting in the type of corner at the end of the first traversing block being different from that at the start point of the second block, the above error message is issued.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	This alarm occurs when an axis that is required for tool radius compensation is not a geometry axis. With CUT2DF, the axis can be a positioning axis perpendicular to the machining plane; with all other types of compensation (CUT2DF, CUT3DC, CUT3DF, CUT3DFF), all geometry axes must be operated as such.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program. On selection of G41/42, the axes involved must be known as GEOAX in the channel. It is possible by programming GEOAX() or G91 G0 X0 Y0 in the block prior to G41/42.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The maximum permissible number of blocks with active compensation suppression with tool radius compensation is limited by MD20252 $MC_CUTCOM_MAXNUM_SUPPR_BLOCKS.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.

Remedy:	- Modify part program. - Modify machine data. - Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the SD42480 $SC_STOP_CUTCOM_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).
Reaction:	Alarm display. NC Stop on alarm at block end.
Remedy:	- Continue machining with CANCEL and Start. - Modify part program. - Set SD42480 $SC_STOP_CUTCOM_STOPRE to FALSE.
Program Continuation:	Clear alarm with the Delete key or NC START.

10780   [Channel %1:] Block %2 impermissible change of a turning or grinding tool with active tool radius compensation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A tool change on which the edge offset (difference between edge center and edge reference point) changes, is only permissible in straight and polynomial blocks. It is impermissible in circular blocks, involute blocks and in blocks including rational polynomials with maximum permissible numerator and denominator degrees.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	- Continue machining with CANCEL and Start. - Modify part program. - Set SD42480 $SC_STOP_CUTCOM_STOPRE to FALSE.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10781   [Channel %1:] Block %2 illegal orientation of involute with tool radius compensation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Tool radius compensation is possible for involutes only if the compensation plane matches the involute plane.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10782   **[Channel %1:] Block %2 illegal curve type with tool radius compensation**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm occurs, if an attempt is made to apply the tool radius compensation to a curve type for which this function is not implemented. The only cause at present: Involute with 3D tool radius compensation.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When activating the tool radius compensation with constraint surface, an illegal tool type is active. Only cutting tools of the tool types 1 to 399 are admitted with the following exceptions: - 111 ball end milling cutter - 155 torus milling cutter - 156 torus milling cutter - 157 torus milling cutter
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Use another tool.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The active plane was changed between the first and second subblock when programming two straight lines with angle parameters.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10791   **[Channel %1:] Block %2 invalid angle during linear programming**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	No intermediate point was found when programming a contour consisting of two straight lines and an angle specification.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set.

Alarm display.
NC Stop on alarm at block end.

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

10792   [Channel %1:] Block %2 illegal interpolation type during linear programming with angles

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Only spline or linear interpolation is permitted for programming two straight lines with angle specification. Circular or polynomial interpolation is not allowed.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10793   [Channel %1:] Block %2 second block missing during linear programming with angles

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The second block is missing during programming of two straight lines with angle specification. This situation only occurs if the first subblock is also the last block of a program, or if the first subblock is followed by a block with a preprocessor stop.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10794   [Channel %1:] Block %2 angle specification missing in 2nd block during linear interpolation with angles

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The angle is missing from the second block during programming of two straight lines with angle specification. This error can only occur if an angle was programmed in the preceding block, but no axis of the active plane was programmed in that block. The cause of the error may therefore also have been the intention to program a single straight line with an angle in the previous block. In this case, exactly one axis of the active plane must be programmed.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10795   [Channel %1:] Block %2 end point specification during angle programming contradictory

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	During programming of a straight line, both positions of the active plane and an angle were specified (the position of the end point is over-specified), or the position of the programmed coordinate cannot be reached with the specified angle. If a contour consisting of two straight lines is to be programmed with angles, it is possible to specify the two axis positions of the plane and an angle in the second block. The error can also occur if, due to a programming error, the preceding block cannot be interpreted as the first subblock of such a contour. A block is interpreted as the first block of a two-block contour if an angle, but not an axis of the active plane, was programmed, and if the block is not already the second block of a contour.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	With an active transformation or a frame with a rotation component the geometry axes are needed for block preparation. If a geometry axis has previously been traversed as positioning axis, it retains its status of "positioning axis" until it is again programmed as a geometry axis. Because of the POSA motion beyond block boundaries, it is not possible to identify in the preprocessing run whether the axis has already reached its target position when the block is executed. This is, however, an unconditional requirement for calculating the ROT component of the frame or of the transformation. If geometry axes are used as positioning axes, then: 1. No rotation may be specified in the current overall frame. 2. No transformation may be selected.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	After selecting transformation or frame, reprogram the geometry axis now operating as positioning axis (e.g. with WAITP) in order to revert the status to "geometry axis.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10805   [Channel %1:] Block %2 repositioning after switch of geometry axes or transformation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the asynchronous subroutine the assignment of geometry axes to channel axes was changed or the active transformation modified.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10810 [Channel %1:] Block %2 master spindle not defined

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The function "Revolutional feedrate" (with G95 or G96), or "Rigid tapping" (with G331/G332) has been programmed, although no master spindle is defined from which the speed could be derived. For the definition the MD 20090 $MC_SPIND_DEF_MASTER_SPIND is available for the default or the keyword SETMS in the part program, thus allowing each spindle of the channel to be redefined as master spindle.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Preset the master spindle with MD20090 $MC_SPIND_DEF_MASTER_SPIND[n]=m (n ... channel index, m ... spindel no.) or define it with an identifier in an NC part program before a G function that requires a master spindle is programmed. The machine axis that is to be operated as a spindle must be equipped in MD35000 $MA_SPIND_ASSIGN_TO_MACHAX[n]=m (n ... machine axis index, m ... spindle no.) with a spindle number. Additionally, the MD20070 $MC_AXCONF_MACHAX_USED[n]=m (n ... channel axis index, m ... machine axis index) must be used to assign it to a channel (channel axis index 1 or 2).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	Revolutional feed has been programmed for contouring and synchronous axes or for an axis/spindle. However, the rotary axis/spindle from which the feed is to be deduced is not available.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Correct part program or set the SD43300 $SA_ASSIGN_FEED_PER_REV_SOURCE correctly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Cause: A traversing velocity has not been programmed for the displayed traversing block. Feed F or FZ: With the traversing velocity defined by feed F or FZ, F or FZ was not reprogrammed after the feed type changed, for example linear feed G94 after revolutional feedrate G95 F or G95 FZ. Modal feed FRCM: With modal traversing velocity FRCM defined for rounding RND or chamfering CHF, feed FRCM was not reprogrammed after the feed type changed, for example linear feed G94 after revolutional feedrate G95, or revolutional feedrate G95 F after tooth feedrate G95 FZ. Note: Feed FRCM also has to be reprogrammed when the feed type changes if the current traversing block does not contain chamfering CHF or rounding RND, but the feed FRCM was programmed active, that is unequal to 0, before the feed type changed.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.

Remedy:	Program feedrate in accordance with the interpolation type. - G93: The feedrate is specified as a time-reciprocal value under address F in [rev/min]. - G94 and G97: The feedrate is programmed under address F in [mm/min] or [m/min]. - G95: The feedrate is programmed as revolutional feedrate under address F in [mm/revolution] or under the address FZ in [mm/tooth]. - G96: The feedrate is programmed as cutting rate under address S in [m/min]. It is derived from the current spindle speed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10861   [Channel %1:] Block %3 velocity of positioning axis %2 is zero

Parameters:	%1 = Channel number %2 = Axis %3 = Block number, label
Definitions:	No axis velocity has been programmed and the positioning velocity set in the machine data is zero.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Enter a different velocity in MD32060 $MA_POS_AX_VELO.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10862   [Channel %1:] Block %2 master spindle also used as path axis

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A contour has been programmed that also includes the master spindle as contouring axis. However, the velocity of the contour is derived from the rotational speed of the master spindle (e.g. G95).
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Modify the program so that no reference is possible to the program itself.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10865   [Channel %1:] Block %2 FZ active, but no tool offset, tool %3

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Tool
Definitions:	For the displayed traversing block a tooth feedrate is active, but no tool offset. Traversing can be performed after the error has been acknowledged. For calculation of the effective feedrate one tooth per revolution will be assumed.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Check the NC program for correct tool selection and correct it, if required; then continue the program with NC start. Or: Continue the NC program with NC start. For calculation of the effective feedrate one tooth per revolution is assumed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

10866 [Channel %1:] Block %2 FZ is active, but the number of teeth of the active D number %4 of tool %3 is zero.

Parameters:	%1 = Channel number %2 = Block number, label %3 = Identifier %4 = D number
Definitions:	Tooth feedrate is active for the displayed traversing block, but a D number of 0 is selected with $TC_DPNT (number of teeth). Traversing can be performed after acknowledgement of the error. For calculation of the effective feedrate one tooth per revolution is assumed.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Check the NC program for correct tool selection and correct it, if required; then continue the NC program with NC start. Or: Continue the NC program with NC start. The feedrate will be calculated assuming 1 tooth.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Constant cutting speed was selected although no transverse axis was applied as reference axis for constant cutting speed or assigned through SCC[AX]. Constant cutting speed can be activated as follows: - Basic position G96, G961 or G962 of G group 29 during booting - Programming of G96, G961 or G962 A reference axis for G96, G961 or G962 can be applied as a transverse axis in MD20100 $MC_DIAMETER_AX_DEF or defined through the instruction SCC[AX].
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check MD20100 $MC_DIAMETER_AX_DEF. Before programming G96, G961 or G962 a transverse axis must be defined as a reference axis for constant cutting speed via MD20100 $MC_DIAMETER_AX_DEF or SCC[AX].
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Between 2 blocks containing contour elements and which are to be joined with a chamfer or a radius (CHF, RND), more blocks without contour information have been programmed than provided for in the MD20200 $MC_CHFRND_MAXNUM_DUMMY_BLOCKS.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.

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

10881   [Channel %1:] Block %2 overflow of local block buffer in the case of chamfers or radii

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Between 2 blocks containing the contour elements and to be joined with a chamfer or a radius (CHF, RND), so many dummy blocks have been programmed without contour information that the internal buffer is too small.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Modify part program such that the number of dummy blocks is reduced.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	No chamfer or radius has been inserted between 2 linear or circle contours (edge breaking) because: There is no straight line or circle contour in the plane There is a movement outside the plane A plane change has taken place The permissible number of empty blocks without traversing information (dummy blocks) has been exceeded.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Correct the part program according to the above error description or change the number of dummy blocks in the channel-specific MD20200 $MC_CHFRND_MAXNUM_DUMMY_BLOCKS to comply with the maximum number allowed for in the program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm is output, if at least one of the relevant blocks when inserting chamfers or radii is so short, that the contour element to be inserted must be reduced against its originally programmed value. The alarm occurs only if bit 4 is set in the MD11411 $MN_ENABLE_ALARM_MASK. Otherwise, the chamfer or radius is adapted without an alarm being output.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify NC program of continue program without modifications after CANCEL and Start or with Start alone.
Program Continuation:	Clear alarm with the Delete key or NC START.

10900   **[Channel %1:] Block %2 no S value programmed for constant cutting speed**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If G96 is active, the constant cutting speed under address S is missing.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Program constant cutting speed under S in [m/min] or deselect the function G96. For example, with G97 the previous feed is retained but the spindle continues to rotate at the current speed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The given feed profile is in part negative. However, negative path feed is not allowed. The feed profile is ignored. The specified feed block end value is taken when traversing over the entire block.
Reaction:	Local alarm reaction. Alarm display.
Remedy:	No action is usually necessary. The alarm message indicates an error in the programming, however, and this should be corrected.
Program Continuation:	Clear alarm with the Delete key or NC START.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The following types of interpolation are allowed in the contour program for stock removal: G00, G01, G02, G03, CIP, CT
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	In the contour subroutine, program only path elements that consist of straight lines and arcs.
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The following errors occurred in the subroutine for the contour during stock removal: - Full circle - Overlapping contour elements - Wrong start position
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	The errors listed above must be corrected in the subroutine for the stock removal contour.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10932	**[Channel %1:] Block %2 preparation of contour has been restarted**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The first contour preparation/contour decoding run must be terminated with EXECUTE.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Program the keyword EXECUTE to terminate the contour preparation in the part program before again calling up contour segmentation (keyword CONTPRON).
Program Continuation:	Clear alarm with the RESET key. Restart part program

10933	**[Channel %1:] Block %2 contour programm does not contain enough contour blocks**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The contour program contains: - Less than 3 contour blocks with CONTPRON - No contour blocks with CONTDCON
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Increase the size of the program with the stock removal contour to include at least 3 NC blocks with movements in both axes of the current machining plane.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10934	**[Channel %1:] Block %2 array for contour segmentation is set too small**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	During contour segmentation (activated with the keyword CONTPRON), the field for the contour table has been detected as too small. For every permissible contour element (circle or straight line) there must be a row in the contour table.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Base the definition of the field variables of the contour table on the contour elements to be expected. The contour segmentation divides some NC blocks into as many as 3 machining cuts. Example: N100 DEF TABNAME_1 [30, 11] Field variables for the contour table provide for 30 machining cuts. The number of columns (11) is a fixed quantity.
Program Continuation:	Clear alarm with the RESET key. Restart part program

10962	**[Channel %1:] Block %2 function %3 not possible with path correction**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Funktionsname
Definitions:	With this software release, the specified function can not yet be used together with tool radius compensation. Please modifiy the part program or obtain a higher software version.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.

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

12000 [Channel %1:] Block %2 address %3 programmed repeatedly

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string of the address
Definitions:	Most addresses (address types) may only be programmed once in an NC block, so that the block information remains unambiguous (e.g. X... T... F... etc. - exception: G and M functions).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. - Remove from the NC program addresses that occur more than once (except for those where multiple value assignments are allowed). - Check whether the address (e.g. the axis name) is specified via a user-defined variable (this may not be easy to see if allocation of the axis name to the variable is performed in the program through computational operations only).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string of the address
Definitions:	The number of times each address type may occur in a NC block is defined internally (for instance, all axes together form one address type to which a block limit also applies).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECTION. The correction pointer positions on the incorrect block. The program information must be split up over several blocks (but make sure that the functions are of the non-modal type!).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12020 [Channel %1:] Block %2 illegal address modification

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Valid address types are 'IC', 'AC', 'DC', 'CIC', 'CAC', 'ACN', 'ACP', 'CACN', 'CACP'. Not each of these address modifications can be used for each address type. The Programming Guide specifies which of these can be used for the various address types. If this address modification is applied to address types that are not allowed, then the alarm is generated, e.g.: N10 G02 X50 Y60 I=DC(20) J30 F100 interpolation parameters with DC.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Key: Press the NC STOP key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer is then positioned on the incorrect block. Apply non-modal address modifications only for permissible addresses, in accordance with the Programming Manual.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string in the block
Definitions:	Some keywords require that the data in their parameters be written in variables of the type "AXIS". For example, in the keyword PO the axis identifier must be specified in the parenthesized expression, and it must be defined as a variable of the AXIS type. With the following keywords only parameters of the AXIS type are possible: AX[..], FA[..], FD[..], FL[..], IP[..], OVRA[..], PO[..], POS[..], POSA[..] Example: N5 DEF INT ZUSTELL=Z1 incorrect, this does not specify an axis identifier but the number 26 161 N5 DEF AXIS ZUSTELL=Z1 correct : N10 POLY PO[X]=(0.1,0.2,0.3) PO[Y]=(22,33,44) &PO[INFEED]=(1,2,3)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the part program in accordance with the instructions given in the Programming Guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = DIN address in the source text block
Definitions:	The name of the DIN address (e.g. X, U, X1) is not defined in the control. In addition to the fixed DIN addresses, the control also has variable addresses. Refer to "Variable addresses" in the Programming Guide. The names of these addresses can be altered by machine data. e.g.: DIN identifier -> Configured identifier G01 -> LINE, G04 -> WAIT ...
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Study the Programming Guide and the machine data with respect to the addresses actually configured and their significance and correct the DIN block accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The G functions that can be used in the part program are divided into groups that are syntax defining or non-syntax defining. Only one G function may be programmed from each G group. The functions within a group are mutually preclusive. The alarm refers only to the non-syntax defining G functions. If several G functions from these groups are called in one NC block, the last of these in a group is active in each case (the previous ones are ignored). Syntax defining G functions: 1. to 4th G group Non-syntax defining G functions: 5. to nth G group
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. No remedy is required. You should, however, check whether the G function last programmed really is the one required.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12070 **[Channel %1:] Block %2 too many syntax-defining G functions**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Syntax defining G functions determine the structure of the part program block and the addresses contained in it. Only one syntax defining G function may be programmed in each NC block. The G functions in the 1st to 4th G group are syntax defining.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Analyze NC block and distribute the G functions over several NC blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12080 **[Channel %1:] Block %2 syntax error in text %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source text area
Definitions:	At the text position shown, the grammar in the block is incorrect. The precise reason for this error cannot be specified in more detail because there are too many possibilities. Example 1: N10 IF GOTOF ... ; the condition for the jump is missing! Example 2: N10 DEF INT VARI=5 N11 X VARI ; the operation is missing for the X and VARI variables
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Analyze the block and correct it in accordance with the syntax rules given in the Programming Guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12090 **[Channel %1:] Block %2 unexpected parameter %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Disallowed parameters in the text
Definitions:	The programmed function has been predefined; no parameters are allowed in its call. The first unexpected parameter is displayed. Example: On calling the predefined subroutine TRAFOF (switching off a transformation) parameters have been transferred (one or more).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Program function without parameter transfer.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12100 **[Channel %1:] Block %2 number of passes %3 not permissible**

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

Definitions:	The subroutines called with MCALL are modal, i.e. after each block with positional information a routine run is automatically performed once. For this reason, programming of the number of passes under address P is not allowed. The modal call is effective until another MCALL is programmed, either with a new subroutine name or without (delete function).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Program the subroutine call MCALL without number of passes.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12110   [Channel %1:] Block %2 block syntax cannot be interpreted

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The addresses programmed in the block are not permissible together with the valid syntax defining G function, e.g. G1 I10 X20 Y30 F1000. An interpolation parameter must not be programmed in the linear block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check the block structure and correct in accordance with the programming requirements.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The G function programmed in this block must be alone in the block. No general addresses or synchronous actions may occur in the same block. These G functions are: G25, G26: Working area and spindle speed limitation G110, G111, G112: Pole programming with polar coordinates G92: Spindle speed limitation with v constant STARTFIFO, STOPFIFO: Control of preprocessing buffer E.g. G4 F1000 M100: no M function allowed in the G4 block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program G function by itself in the block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12140   [Channel %1:] Block %2 functionality %3 not implemented

Parameters:	%1 = Channel number %2 = Block number, label %3 = Software construct in the source text
Definitions:	In the full configuration of the control functions are possible that are not yet implemented in the current version.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. The displayed function must be removed from the program.

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

12150 [Channel %1:] Block %2 operation %3 not compatible with data type

Parameters:	%1 = Channel number %2 = Block number, label %3 = String (violating operator)
Definitions:	The data types are not compatible with the required operation (within an arithmetic expression or in a value assignment). Example 1: Arithmetic operation N10 DEF INT OTTO N11 DEF STRING[17] ANNA N12 DEF INT MAX : N50 MAX = OTTO + ANNA Example 2: Value assignment N10 DEF AXIS DRILL N11 DEF INT OTTO : N50 OTTO = DRILL
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Alter the definition of the variables used such that the required operations can be executed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Impermissible value
Definitions:	The programmed constant or the variable lies beyond the value range that has previously been established by the definition of the data type. An initial value in a DEF or REDEF instruction lies beyond the upper (ULI) or lower (LLI) limit values that have been programmed or already exist in the DEF instruction.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct value of the constant or adapt data type. If the value for an integer constant is too great, it can be specified as real constant by adding a decimal point. Example: R1 = 9 876 543 210 Correct: R1 = 9 876 543 210. Value range INTEGER: $+/-(2^{**}31 - 1)$ Value range REAL: $+/-(10^{**}-300 .. 10^{**}+300)$
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12161 [Channel %1:] Block %2 Error on defining the limit %3

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

Definitions:	The alarm may have the following causes.
	  - During definition (DEF) or redefinition (REDEF) of a variable's limits, a value was entered for the upper limit that is smaller than that for the lower limit.
	  - A limit for a variable type was programmed that is not of type CHAR, INT or REAL.

	- A limit value of type CHAR was programmed for a variable with the data type INT or REAL.
	- A string (more than one character) was programmed for one of the limits.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	If the alarm occurs in the part program (DEF instruction), press the NC Stop key and select the function "Compensation block" using PROGRAM CORRECT softkey. The cursor is placed on the incorrect block.
	Adjust the limit values afterwards or completely remove the limit programming in the case of an incorrect data type.
	If the alarm occurs on compiling a GUD or ACCESS file, correct the GUD or ACCESS definition file (DEF file).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12162   [Channel %1:] Block %2 Physical unit not allowed

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	In a DEF or REDEF instruction, a physical unit may only be defined for variables of data type INT or REAL.
	Furthermore, only the following values may be programmed for the physical unit:

0   No physical unit
1   Linear or angular position, dependent upon axis type
2   Linear position [mm ; inch]
3   Angular position [degrees]
4   Linear or angular velocity, dependent upon axis type
5   Linear velocity [mm/min]
6   Angular velocity [rev/min]
7   Linear or angular acceleration, dependent upon axis type
8   Linear accel. [m/s² ; inch/s²]
9   Angular accel. [rev/s²]
10   Linear or angular jerk
11   Linear jerk [m/s³ ; inch/s³]
12   Angular jerk [rev/s³]
13   Time [s]
14   Position controller gain [16.667/s]
15   Revolutional feedrate [mm/rev ; inch/rev]
16   Unit for temperature compensation values, dependent upon axis type
18   Force [N]
19   Weight [kg]
20   Moment of inertia [kgm²]
21   Per cent
22   Frequency [Hz]
23   Voltage [V]
24   Current [A]
25   Temperature [degrees Celsius]
26   Angle [degrees]
27   KV [1000/min]
28   Linear or angluar position [mm|deg or inch|deg]
29   Cutting velocity [m/min; feet/min]
30   Peripheral velocity [m/s; feet/s]
31   Resistance [ohms]
32   Inductance [mH]
33   Torque [Nm]
34   Torque constant [Nm/A]
35   Current controller gain [V/A]
36   Speed controller gain [Nm/rad s 1]
37   Speed [rev/min]
42   Power [kW]
43   Low power [µA]
46   Low torque [µNm]
48   Per mill HZ_PER_SEC = 49, [Hz/s]
65   Flow [l/min]
66   Pressure [bar]
67   Volume [cm³]

68   Controlled system gain [mm/Vmin]
69   Controlled system gain force controller [N/V]
155 Thread pitch [mm/rev; inch/rev]
156 Thread pitch change [mm/rev² ; inch/rev²]

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	If the alarm occurs in the part program (DEF instruction), press the NC Stop key and select the function "Compensation block" using PROGRAM CORRECT softkey. The cursor is placed on the incorrect block. In the compensation block the data type can now be adjusted in the DEF instruction, or the physical unit (PHU xy) must be removed. If the alarm occurs on compiling a GUD or ACCESS file, correct the GUD or ACCESS definition file (DEF file).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12163   **[Channel %1:] Block %2 Change of access protection not allowed**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Changing the access rights for system variables (with REDEF) is not allowed in GUD files. They can only be changed in the ACCESS files (_N_SYSACCESS_DEF, _N_SACCESS_DEF, _N_MACCESS_DEF and _N_UACCESS_DEF).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Cut the REDEF instruction from the GUD file and paste it into one of the ACCESS files.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12164   **[Channel %1:] Block %2 access protection programmed more than once %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Duplicate protection level programming
Definitions:	The language commands APW and APR are used to program access protection for access to both the part program and the OPI. APWP and APRP define access protection from the part program; APWB and APRB define access protection via the OPI. Programming APW in the same block together with APWP or APWB or programming APR in the same block together with APRP or APRB will lead to a conflict, as the protection level assignment is no longer unique.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	If access protection in the part program and on the OPI needs to be set to diffferent levels, only the language commands APWP, APWB, APRP and APRB may be used. If access protection is to be the same in the part program and on the OPI, it can also be programmed with APW or APR, although in this case the APWP and APWB or APRP und APRB commands must not be programmed in the same block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Symbol in block
Definitions:	The symbol shown in the error message has already been defined in the active part program. Note that user-defined identifiers may occur more than once if the multiple definition occurs in other (sub)programs, i.e. local variables may be redefined with the same name if the program has been exited (subprograms) or has already been concluded. This applies both to user-defined symbols (labels, variables) and to machine data (axes, DIN addresses and G functions).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	The symbol already known to data management is displayed. This symbol must be looked for in the definition part of the current program using the program editor. The 1st or 2nd symbol must be given a different name.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12180 [Channel %1:] Block %2 illegal chaining of operators %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Chained operators
Definitions:	Operator chaining means the writing in sequence of binary and unary operators without using any form of parentheses. Example: N10 ERG = VARA - (- VARB) ; correct notation N10 ERG = VARA - - VARB ; error!
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Formulate the expression correctly and unambiguously making use of parentheses. This improves clarity and readability of the program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12185 [Channel %1:] Block %2 a bit combination with %3 is not permitted

Parameters:	%1 = Channel number %2 = Block number, label %3 = Address name
Definitions:	A bit combination is not possible with the assignment to this address. Bit combinations are permitted only for coupling adresses (CPMBRAKE, CPMVDI and CPMAL).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Change the part rogram. If the data type of the address permits a bit combination, write the value of the address in a variable, make a bit combination with the variable, and assign the variable to the address.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12190 [Channel %1:] Block %2 variable of type ARRAY has too many dimensions

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Array with variables of type STRING may be no more than 1-dimensional, and with all other variables no more than 2-dimensional.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the array definition, with multi-dimensional arrays define a second 2-dimensional array if necessary and operate it with the same field index.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

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

Definitions:	The symbol to be created with the DEF instruction cannot be created because: - it has already been defined (e.g. as variable or function) - the internal memory location is no longer sufficient (e.g. with large arrays)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Make the following checks: - Check with the text editor whether the name to be allocated in the active program cycle (main program and called subprograms) has already been used. - Estimate the memory requirements for the symbols already defined and reduce these if necessary by using fewer global and more local variables.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12205 [Channel %1:] Block %2 area specification missing for GUD area

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The area specification (NCK or CHAN) was not programmed in the definition instruction for a GUD variable.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Complete the area specification for the GUD variable definition in the GUD definition file. The definition of a GUD variable must conform to the following syntax: DEF <Area> <Data type> <Variable name> e.g. DEF NCK INT intVar1 DEF CHAN REAL realVar1
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = String in the source block
Definitions:	- In the definition of a STRING type variable, an attempt has been made to initialize more than 200 characters. - In an allocation, it has been found that the string does not fit the given variable. - In synchronized actions, a string with more than 31 characters has been programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the "Correction block" function with the PROGRAM CORRECT softkey. The cursor is positioned on the incorrect block. - Select a shorter string or divide the character string into 2 strings. - Define a longer string variable. - Restrict the string to 31 characters.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12220 [Channel %1:] Block %2 binary constant %3 in string too long

Parameters:	%1 = Channel number %2 = Block number, label %3 = Binary constant
Definitions:	When initializing or allocating the value of a variable of type STRING more than 8 bits have been found as binary constant. DEF STRING[8] OTTO = "ABC'H55"B000011111'DEF"
Reaction:	Correction block is reorganized. Interface signals are set.

	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. In the window for the alarm message, the first characters of the binary constant are always displayed although the surplus bit might not be located until further on. Therefore, the complete binary constant must always be checked for an incorrect value.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12230 [Channel %1:] Block %2 hexadecimal constant %3 in string too long

Parameters:	%1 = Channel number %2 = Block number, label %3 = Hexadecimal constant
Definitions:	A string can also contain bytes that do not correspond to a character that can be entered or one that is available on a keyboard with a minimized number of keys. These characters can be input as binary or hexadecimal constants. They may occupy up to 1 byte each only - therefore be < 256, e.g. N10 DEF STRING[2] OTTO=" 'HCA' 'HFE' "
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. In the window for the alarm message, the first characters of the hexadecimal constant are always displayed although the surplus bit might not be located until further on. Therefore, the complete hexadecimal constant must always be checked for an incorrect value.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12240 [Channel %1:] Block %2 tool orientation %3 defined repeatedly

Parameters:	%1 = Channel number %2 = Block number, label %3 = Text
Definitions:	Only 1 tool orientation can be programmed per DIN block. This can either be defined via the 3 Euler angles, or the end points of the axes, or through direction vectors.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Since the tool orientation can be set in 3 different ways, the most advantageous should be selected. For this type of specification, the addresses and value assignments must be programmed and all other orientation parameters must be removed. Axis end points (additional axes): A, B, C axis identifiers Euler angles: A2, B2, C2 Direction vectors: A3, B3, C3
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12250 [Channel %1:] Block %2 nested macro %3 not possible

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The macro technique supplies a 1-line instruction or series of instructions with a new identifier by means of the keyword DEFINE. No further macro may be contained in the string of instructions (nesting). Example: N10 DEFINE MACRO1 AS G01 G91 X123 MACRO2 F100
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Nested macros must be replaced by the full program information.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12260 [Channel %1:] Block %2 too many initialization values specified %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	In the initialization of an array (array definition and value assignments to individual array elements) there are more initialization values than array elements. Example: N10 DEF INT OTTO[2,3]=(..., ..., {more than 6 values})
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check the NC program to establish whether: 1. During array definition the number of array elements (n,m) was indicated correctly (DEF INT FIELDNAME[n,m] e.g. an array with 2 lines and 3 columns: n=2, m=3). 2. During initialization the value assignments have been made correctly (values of the individual field elements separated by comma, decimal point for variables of the type REAL).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12261 [Channel %1:] Block %2 initialization of %3 not allowed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	Frame type variables cannot be initialized in the definition. Example: DEF FRAME LOCFRAME = CTRANS(X,200) Equally, no default values can be programmed for axes in the program run during field initialization via SET. A REDEF instruction with PRLOC is only permitted for setting data, but not for machine data or variables.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	IPerform initialization in separate block in the execution part of the program: DEF FRAME LOCFRAME LOCFRAME = CTRANS(X,200) When using for axis variables: Replace DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = SET (X, , Y) by: DEF AXIS AXIS_VAR [10] AXIS_VAR [5] = X AXIS_VAR [7] = Y If REDEF ... INIRE, INIPO, INICF, PRLOC changes the behavior of a GUD, LUD etc., then the MD11270 $MN_DEFAULT_VALUES_MEM_MASK must equal 1.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12270 [Channel %1:] Block %2 macro identifier %3 already defined

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string macro name
Definitions:	The name of the macro to be selected by the instruction DEFINE is already defined in the control as: Macro name Keyword Variable Configured identifier.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Select DEFINE instruction with another macro name.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12280 [Channel %1:] Block %2 maximum macro length %3 exceeded

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The string of instructions on the right side of the macro is limited to 256 characters. If an attempt is made to define a longer character string under one macro (possible only through V.24 input of NC blocks, because communication between operator panel and NCK is limited to a block length of 242 characters), an alarm is displayed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Divide the functions defined under the macro into 2 macros.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12290 [Channel %1:] Block %2 arithmetic variable %3 not defined

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string arithmetic variable
Definitions:	Only the R parameters are predefined as arithmetic variables. All other arithmetic variables must be defined with the DEF instruction before being used. The number of arithmetic parameters is defined via machine data. The names must be unambiguous and may not be repeated in the control (exception: local variables).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Define the required variable in the definition part of the program (possibly in the calling program if it is to be a global variable).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12300 [Channel %1:] Block %2 call-by-reference parameter missing on subroutine call %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	In the subroutine definition, a formal REF parameter (call-by-reference parameter) has been specified with no actual parameter assigned to it. The assignment takes place in the subroutine call on the basis of the position of the variable name and not on the basis of the name! Example: Subroutine: (2 call-by-value parameters X and Y, 1 call-by-reference parameter Z) PROC XYZ (INT X, INT Y, VAR INT Z) : M17 ENDPROC Main program: N10 DEF INT X N11 DEF INT Y N11 DEF INT Z

	: N50 XYZ (X, Y); REF parameter Z missing or N50 XYZ (X, Z) ; REF parameter Z missing!
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Assign a variable to all REF parameters (call-by-reference parameters) of the subroutine when calling. No variable must be assigned to "normal" formal parameters (call-by-value parameters), as these are defaulted with 0.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	When calling the subroutine, an AXIS parameter is missing which, according to the EXTERN declaration, should be present. With the EXTERN instruction, user-defined subroutines (procedures) are made "known" that have a parameter transfer. Procedures without parameter transfer require no EXTERN declaration. Example: Subroutine XYZ (with the formal parameters): PROC XYZ (INT X, VAR INT Y, AXIS A, AXIS B) EXTERN instruction (with variable types): EXTERN XYZ (INT, VAR INT, AXIS, AXIS) Subroutine call (with actual parameters): N10 XYZ (, Y1, R_TISCH) Variable X is defaulted with value 0 Variable Y is supplied with the value of the variable Y1 and returns the results to the calling program after the subroutine run Variable A is supplied with the axis in R_TISCH Variable B missing!
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Program the missing AXIS parameter in the call.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12320 [Channel %1:] Block %2 parameter %3 is no variable

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	A constant or the result of a mathematical expression has been assigned to a REF parameter instead of a variable at the time of the subroutine call, even though only variable identifiers are allowed. Examples: N10 XYZ (NAME_1, 10, OTTO) or N10 XYZ (NAME_1, 5 + ANNA, OTTO)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remove the constant or the mathematical expression from the NC block.

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

12330 [Channel %1:] Block %2 type of parameter %3 incorrect

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	When calling a procedure (a subroutine) it is found that the type of the actual parameter cannot be converted into the type of the formal parameter. There are two possible cases: - Call-by-reference parameter: Actual parameter and formal parameter must be of precisely the same type, e.g. STRING, STRING. - Call-by-value parameter: Actual parameter and formal parameter can in principle be different providing conversion is basically possible. In the present case, however, the types are generally not compatible, e.g. STRING -> REAL. Overview of type conversions: - from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: - - from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: - - from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: - - from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: - - from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: - - from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: - - from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes 1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE. 2) String length 0 => FALSE, otherwise TRUE. *) At type conversion from REAL to INT fractional values that are >=0.5 are rounded up, others are rounded down.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check transfer parameters of the subroutine call and define the application accordingly as call-by-value or call-by-reference-parameter.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	When calling a function or a procedure (predefined or user-defined) more parameters were transferred than defined. Predefined functions and procedures: The number of parameters has been set permanently in the NCK. User-defined functions and procedures: The number of parameters is established by type and name in the definition.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check whether the correct procedure/function has been called. Program the number of parameters in accordance with the procedure/function.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12350 [Channel %1:] Block %2 parameter %3 no longer possible

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

Definitions:	An attempt has been made to transfer actual parameters although axis parameters located before them have not been assigned. For procedure or function calls, the assignment of parameters that are no longer required can be omitted if no more parameters are to be transferred subsequently. Example: N10 FGROUP(X, Y, Z, A, B); a max. of 8 axes possible The following call-by-value parameters would then be initialized with zero because the space-dependent assignment has been lost on account of the missing axis parameters. Axes that can be omitted and following parameters do not occur in the predefined procedures and functions.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. In predefined procedures and functions either remove the following parameters or transfer any preceding axis parameters. In user-defined procedures and functions, parameter transfer must be programmed in accordance with the instructions given in the machine manufacturer's programming guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The following possibilities of error must be checked: - The current parameter is an array, but the formal parameter is a variable - The current parameter is a variable, but the formal parameter is an array - The current and formal parameters are arrays, but not with the dimensions to be defined.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the NC part program in accordance with the cause of error as listed above.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	A variable has been initialized with a value range outside an initialization block. The definition of program-global variables is allowed only in special initialization blocks. These variables can be initialized with a value range.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remove the value range specification (begins with the keyword OF) or define the variable as a global variable in the initialization block and initialize it with a value range.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The data definitions in this block cannot be processed because the maximum available memory for creating the data has been filled, or because the data block cannot accommodate any further data. The alarm can also occur if several subroutine calls are executed in sequence and no block with an effect on the machine is generated (motion, dwell, M function).

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Reduce the number of variables, reduce the size of arrays, or increase the capacity of the data management system. - If new macro definitions are to be introduced -> Increase MD18160 $MN_MM_NUM_USER_MACROS - If new GUD definitions are to be introduced -> Check MD18150 $MN_MM_GUD_VALUES_MEM, MD18130 $MN_MM_NUM_GUD_NAMES_CHAN, MD18120 $MN_MM_NUM_GUD_NAMES_NCK - If the error occurs while executing an NC program with LUD definitions or when using cycle programs (the parameters count as LUD variable of the cycle program), the following machine data must be checked: MD28040 $MC_MM_LUD_VALUES_MEM, MD18242 $MN_MM_MAX_SIZE_OF_LUD_VALUE, MD18260 $MN_MM_LUD_HASH_TABLE_SIZE, MD28020 $MC_MM_NUM_LUD_NAMES_TOTAL, MD28010 $MC_MM_NUM_REORG_LUD_MODULES
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	During initialization, a value has been assigned to a variable that does not correspond to the type of the variable, nor can it be converted to the data type of the variable. Overview of type conversions: - from REAL to REAL: no, INT: yes1), BOOL: yes, CHAR: yes2), STRING: - - from INT to REAL: yes, INT: no, BOOL: yes, CHAR: yes2), STRING: - - from BOOL to REAL: yes, INT: yes, BOOL: no, CHAR: yes, STRING: - - from CHAR to REAL: yes, INT: yes, BOOL: yes, CHAR: no, STRING: yes - from STRING to REAL: -, INT: -, BOOL: yes, CHAR: yes3), STRING: no 1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE. 2) String length 0 => FALSE, otherwise TRUE. 3) If only one character. It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. - Define variable type such that the initialization value can be assigned, or - Select initialization value in accordance with the variable definition.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The following causes are possible: - Impermissible index list; an axis index is missing - Array index does not match the definition of the variables - An attempt was made to access a variable at array initialization via SET or REP; this attempt did not correspond to the standard access. Single character access, partial frame access, omitted indices not possible. A non-existent element was addressed on initializing this array.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
	Array initialization: Check the array index of the addressed element. The 1st array element is given the index [0,0], the 2nd array element [0,1] etc. The right array index (column index) is incremented first.
	In the 2nd row, the 4th element is also addressed with the index [1,3] (the indices start at zero).
	Array definition: Check the size of the array. The1st number indicates the number of elements in the 1st dimension (number of rows), the 2nd number indicates the number of elements in the 2nd dimension (number of columns). An array with 2 rows and 3 columns must be defined by specifying [2,3].
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source string
Definitions:	In assigning a value to an element of an array variable, the array index was specified in a way that is not allowed. Only the following are allowed as array index (in square brackets):
	- Axis identifier, provided the array variable was defined as data type FRAME.
	- Integer values for all other data types.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct indices of the array element with respect to variable definition or define the array variable differently.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12420 [Channel %1:] Block %2 identifier %3 too long

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The symbol to be defined or the specified jump target has a name which is longer than the 32 characters allowed.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. The symbol to be created or the target of program jumps (label) must conform to the system specifications, that means the name must begin with 2 letters (but the 1st sign must not be "$") and may be up to a maximum of 32 characters.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12430 [Channel %1:] Block %2 specified index is invalid

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	In specifying an array index (in the array definition) an index was used that is outside the permissible range.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Specify array index within the permissible range. Value range per array dimension: 1 - 32 767.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12440   **[Channel %1:] Block %2 maximum number of formal arguments exceeded**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the definition of a procedure (a subroutine) or in an EXTERN instruction, more than 127 formal parameters have been specified. Example: PROC ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, FORMPARA128, ...) EXTERN ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, FORMPARA128, ...)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. A check must be made to determine whether all parameters really have to be transferred. If so, the formal parameters can be reduced by using global variables or R parameters, or by grouping together parameters of the same type to form an array and transfer them in this form.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The label of this block already exists. If the NC program is compiled off-line, the entire program is compiled block for block. During this procedure all multiple labels are recognized; this is not always the case with on-line compilation. (Only the actual program run is compiled here, i.e. program branches that are not passed through in this run are disregarded and could therefore contain programming errors).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer is positioned on the block where the displayed label occurs for the 2nd time. Use the editor to search the part program where this label occurs for the 1st time, and change one of the names.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The max. number of variable definitions (GUD, LUD), macro definitions, cycle programs and/or cycle parameters (PROC instruction) that the controller's data management system is able to handle has been exceeded. If this alarm occurs in conjunction with alarm 15175, not enough memory for the preprocessing of the cycle program definitions is available (PROC instruction). If this alarm occurs in conjunction with alarm 15180, then this alarm shows the name of the file (INI or DEF file) causing the error. (For a list of names of INI files and their meaning -> please refer to alarm 6010)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Generally reduce the number of symbols in the affected block (possibly by using the array technique or by using R parameters), or adapt the corresponding machine data (see below). MD28020 $MC_MM_NUM_LUD_NAMES_TOTAL with error in LUD blocks (i.e. if more variable definitions were made in the active part program than allowed by the MD). GUD data blocks can cause errors as part of the 'initial.ini download' process (e.g. in the case of a series start-up) or by selective activation via PI service _N_F_COPY (activate GUD via HMI dialog). If alarm 15180 refers to a GUD definition file, then MD18120 $MN_MM_NUM_GUD_NAMES_NCK or MD18130 $MN_MM_NUM_GUD_NAMES_CHAN is set to a value that is too small.

Macros are loaded during POWER ON/NCK RESET or selectively via PI service _N_F_COPY (activate macro via HMI dialog). If alarm 15180 refers to a macro definition file, then MD18160 $MN_MM_NUM_USER_MACROS is set to a value that is too small.

Cycle program definitions (PROC instruction) are reloaded at each POWER ON/NCK RESET. In case of failure check parameter %3 to find out whether the name of the cycle program has caused the error - in this case, the value of MD18170 $MN_MM_NUM_MAX_FUNC_NAMES should be increased, or whether the name of a cycle call parameter has caused the error - in this case, the value of MD18180 $MN_MM_NUM_MAX_FUNC_PARAM should be increased.

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

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	With indirectly programmed G functions, an invalid or non-allowed group number has been programmed. Allowed group number = 1. and 5 max. number of G groups. In the displayed block, a non-defined G function has been programmed. Only "real" G functions are checked, which begin with the address G, e.g. G555. "Named" G functions such as CSPLINE, BRISK etc. are interpreted as subroutine names.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. You should decide on the basis of the machine manufacturer's programming guide whether or not the displayed G function exists or is available, or whether a standard G function has been reconfigured (or introduced by an OEM). Remove G function from the part program or program function call in accordance with the machine manufacturer's programming guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = G code number
Definitions:	A non-allowed G function number (parameter 3) has been programmed for a G group with indirect G code programming. Only the G function numbers indicated in the Programming Guide "Fundamentals", Section 12.3 "List of G functions/Path conditions" are allowed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12480   [Channel %1:] Block %2 subroutine %3 already defined

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The name used in the PROC or EXTERN instruction has already been defined in another call description (e.g. for cycles). Example: EXTERN CYCLE85 (VAR TYP1, VAR TYP2, ...)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. A program name must be selected that has not yet been used as identifier. (Theoretically, the parameter declaration of the EXTERN instruction could also be adapted to the existing subroutine in order to avoid the alarm output. However, it would have been defined identically twice).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12481 [Channel %1:] Block %2 program attribute %3 not allowed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The attribute used in the PROC instruction is not permitted in the current operating mode. The attribute SAVE, for example, is not allowed in a technology cycle.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press button NC STOP and select the function "Compensation block" using softkey PROGAM CORRECT. The cursor jumps to the incorrect block. Then delete the invalid program attribute.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The desired access authorization, programmed with the keyword REDEF, has not been set. The desired protection level is either beyond the permitted value range or the protection level change is not allowed. (The REDEF instruction is only executable in INITIAL_INI blocks on SINUMERIK 840D, P1 (6/94)). The protection level may be changed only if: 1. The current protection level is equal to or higher than the level originally defined, and 2. The new protection level is to be below the level originally defined. The higher numerical values represent the lower protection levels. The lower 4 levels (from 7 to 4) correspond to the keyswitch positions, and the upper 4 levels are associated with 4 passwords.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. - Use the REDEF instruction only in the INITIAL_INI block - Using the operator panel, set the current protection level to at least the same level as that of the variable with the highest level - Program protection level within the permissible value range - Only program new protection levels that are lower than the old values
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12495 [Channel %1:] Block %2 Change (definition) of data class %3 is not allowed here

Parameters:	%1 = Channel number %2 = Block number, label %3 = Data class
Definitions:	Change of the data class in this ACCESS file or definition in this GUD file (for file name see alarm 15180) not possible. Priority of the new data class may only be smaller or the same as that of the definition file. This means that DCS may only be programmed in SGUD (SACCESS), DCM not in UGUD and GUD9 (UACCESS), DCU not in GUD9. DCI is allowed in all GUD and ACCESS files.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Program the data class of the area permissible for this GUD or ACCESS file.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The displayed keyword may not be used in this type of block and at this location (all files in the NCK are designated as blocks). Block types: Program block Contains a main program or subroutine Data block Contains macro or variable definitions and possibly an M, H or E function Initialization block Contains only selected language elements for data initialization
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remove the displayed language elements (keyword) with its parameters from this block and insert in the block provided for this purpose.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	In the part program, in the machine data file (..._TEA) and in the initialization file (..._INI), no more than 5 machine data may be used per block. Example: N ... N 100 $MN_OVR_FACTOR_FEEDRATE [10] = 15, $MN_OVR_FACTOR_FEEDRATE [11] = 20 N ...
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. - Divide up the part program block into several blocks. - If necessary, use the local variable for storing intermediate results.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12520 [Channel %1:] Block %2 too many tool parameters %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	In the part program, in the tool offset file (..._TOA) and in the initialization file (..._INI), no more than 5 tool offset parameters may be used per block. Example: N ... N 100 $TC_DP1 [5,1] = 130, $TC_DP3 [5,1] = 150.123, $TC_DP4 [5,1] = 223.4, $TC_DP5 [5,1] = 200.12,

	$TC_DP6 [5,1] = 55.02
	N ...
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
	- Divide up the part program block into several blocks.
	- If necessary, use the local variable for storing intermediate results.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source string
Definitions:	In macro definitions, an attempt was made to define a G function with more than 3 decades or an M function with more than 2 decades as identifier of the macro.
	Example:
	_N_UMAC_DEF DEFINE G4444 AS G01 G91 G1234
	DEFINE M333 AS M03 M50 M99
	:
	M17
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
	Modify the macro definition in accordance with the Programming Guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12540   [Channel %1:] Block %2 Block is too long or too complex

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The maximum internal block length after translator processing must not exceed 256 characters. After editing, for example, several macros in the block or a multiple nesting, this limit can be exceeded.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
	Divide up the program block into several subblocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12550   [Channel %1:] Block %2 name %3 not defined or option/function not activated

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source symbol
Definitions:	The identifier displayed has not been defined before being used.
	Macro: The keyword, to be defined with the DEFINE ... AS ... instruction, is missing in one of the files:
	_N_SMAC_DEF
	_N_MMAC_DEF
	_N_UMAC_DEF
	_N_SGUD_DEF
	_N_MGUD_DEF

	_N_UGUD_DEF Variable: DEF instruction is missing Program: PROC declaration is missing The T word cannot be interpreted in ISO mode 2, $MN_EXTERN_DIGITS_TOOL_NO and $MN_EXTERN_DIGITS_OFFSET_NO are 0.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Compensation block" with the softkey PROGRAM CORRECT. The cursor positions itself on the incorrect block. - Correct the name used (writing error) - Check definitions of variables, subroutines and macros - Declare subroutine with EXTERN, load subroutine to SPF-Dir - Check interface definition of subroutine - Check options. See also MD10711 $MN_NC_LANGUAGE_CONFIGURATION.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12552 [Channel %1:] Block %2 tool/magazine OEM parameter not defined. Option not set. Option not set.

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The programmed $TC_... Cx system variable is not known in the control.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Correct the name used (writing error) - $TC_DPCx, $TC_TPCx, $TC_MOPCx, $TC_MAPCx, $TC_MPPCx, $TC_DPCSx, $TC_TPCSx, $TC_MOPCSx, $TC_MAPCSx, $TC_MPPCSx; with x=1,...10 - These are the OEM parameters of the tools magazines, The corresponding machine data value is set to < 10, or the option 'TM OEM parameters' has not been set. - Use correct parameter number, or - if the name cannot be changed - set machine data correction (see MD18096 $MN_MM_NUM_CC_TOA_PARAM, ... MD18206 $MN_MM_NUM_CCS_TOA_PARAM, ...).) - Check the option (machine data are only effective when the option is enabled).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12553 [Channel %1:] Block %2 name %3 option/function is not active

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	The option (if MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 1) or the NC function (if MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 3) related to this language command is not active. But the name of the language command is known. Each programming of this language command is rejected with this alarm.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the "Compensation block" function by pressing the PROGRAM CORRECT softkey. The cursor positions itself on the incorrect block. - Correct the name used (in the case of a typing error). - Activate the NC function (if a language command of an inactive function has been programmed). - Enable the option required (if a language command of a function with a non-enabled option has been programmed). See also MD10711 $MN_NC_LANGUAGE_CONFIGURATION.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12554   **[Channel %1:] Block %2 replacement cycle %3 for the predefined procedure is missing.**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Cycle name
Definitions:	The replacement cycle that is to be called instead of the predefined procedure is not present / unknown in the control.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC stop key and select the "Compensation block" function by pressing the PROGRAM CORRECT softkey. The cursor will position itself in the faulty block. - Correct the name used for the predefined procedure (write error) - Or load the replacement cycle into one of the cycle directories (+ restart) - Or set the machine data bit for the predefined procedure in MD11754 $MN_COUPLE_CYCLE_MASK to 0 so that the predefined procedure is executed again.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12555   **[Channel %1:] Block %2 function not available (identification %3)**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Fine ID
Definitions:	The identifier has not been defined for this system.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC stop key and select the "Compensation block" function by pressing the "Program correct" softkey. The correction indicator will position in the incorrect block. - Correct the name used (write error) - Use a better software system in case of malfunction - Check the definition of variables, subroutines and macros - Declare a subroutine with EXTERNAL; load the subroutine to SPF-Dir - Check the interface definition of the subroutine
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12556   **[Channel %1:] Block %2 name %3 Name is already known**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	The name of the symbol to be created is part of the NC language scope and therefore already known. Although the NC function is not active, this name can no longer be used for GUDs, macros and PROC definitions.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press key NC Stop and select "Correction block" function by pressing softkey "Program correct". The correction indicator is set to the incorrect block. - Correct the name used (typing error) - With MD10711 $MN_NC_LANGUAGE_CONFIGURATION = 2 or 4, only those language commands are created, the option of which has been set or the function of which is active.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12560 [Channel %1:] Block %2 programmed value %3 exceeds allowed limits

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	In a value assignment, the permissible value range of the data type has been exceeded.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Press the NC Stop key and select the function "Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Assign value within the value range of the various data types, or if necessary use another type in order to increase the size of the value range, e.g. INT ->REAL. Value ranges of the various variable types: - REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023) - INT: Property: Integers with signs, value range: +/-(231-1) - BOOL: Property: Truth value TRUE, FALSE, value range: 0,1 - CHAR: Property: 1 ASCII character, value range: 0-255 - STRING: Property: Character string (max. 100 values), value range: 0-255 - AXIS: Property: Axis addresses, value range: Axis names only - FRAME: Property: Geometric information, value range: As for axis paths
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12570 [Channel %1:] Block %2 too many motion synchronous actions in %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	No more than 16 actions are allowed in a block with motion synchronous action.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Reduce the number of programmed actions.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12571 [Channel %1:] Block %2 %3 not permissible for motion synchronous action

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	The predefined subprogram %3 specified here is not allowed in a block with motion synchronous action. It may only be contained in a "normal" block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12572 [Channel %1:] Block %2 %3 only permissible for motion synchronous action

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	The predefined subprogram %3 specified here is only allowed in a block with motion synchronous action. It must not be contained alone in a "normal" block.
Reaction:	Correction block is reorganized.

	Interface signals are set.
	Alarm display.
Remedy:	Modify program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source text area
Definitions:	Call by reference parameters (keyword VAR) are not possible with technology cycles.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct PROC instruction of technology cycle.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12580 [Channel %1:] Block %2 %3 not permissible for assignment in motion synchronous action

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source symbol
Definitions:	The variable displayed must not be written in a motion synchronous action. Only selected variables are permitted here, e.g. DO $AA_IW[X]=10 is not allowed.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Please inform the authorized personnel/service department.
	Modify part program.
	In a motion synchronous action, only certain variables are allowed.
	E.g. $AA_IM, $AC_DTGPB
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12581 [Channel %1:] Block %2 invalid read access to %3 while in motion synchronous action

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Source symbol
Definitions:	In a motion synchronous action, the displayed variable must not be entered as a variable that is to be read online, i.e.
	1. The displayed variable must not be written to the left of the comparison in a motion synchronous action. Only selected variables are permissible, e.g. WHEN $AA_OVR == 100 DO ....
	2. In a motion synchronous action, the displayed variable must not be used as a $$ variable, e.g. WHEN $AA_IM[X] >= $$P_AD[1] DO ... DO $AC_VC = $$P_F
	3. The displayed variable must not be programmed as an online evaluated parameter of a synchronous procedure, e.g. DO SYNFCT(1, $AC_PARAM[0], $SA_OSCILL_REVERSE_POS2[Z])
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12582 [Channel %1:] Block %2 field index %3 incorrect

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	$A or $V variables are assessed in real-time in motion synchronous actions, i.e. in the interpolation cycle. All other variables (e.g. user-defined variables) are still computed at block preparation. It is not permissible to index the index of a variable for block preparation with a real-time variable. Example: DEF INT INPUT[3] WHEN $A_IN[1] == INPUT[$A_INA[1]] DO ... The locally defined variable INPUT must not be indexed with a real-time variable. Program editing: WHEN $A_IN[1] == $AC_MARKER[$A_INA[1]] DO ...
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify program: Use real-time variables.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12583 [Channel %1:] Block %2 variable %3 no system variable

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	In motion synchronous actions, only special system variables are allowed on the left side of the compare operation for the assigned variable as input and result variable of SYNFCT and as input variable for PUTFTOCF. Real-time synchronous access is allowed here. The programmed variable is not a system variable. Example: DEF REAL OTTO, BERTA[2] DO SYNFCT(2,OTTO, $MN_...) ; Local variables or machine data are not allowed as parameter for SYNFCT.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. Local variables or machine data are not allowed as parameters for SYNFCT.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12584 [Channel %1:] Block %2 variable %3 cannot be read synchronously with motion

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	In motion synchronous actions on the left side of the compare operation, only special variables are allowed as input variables of SYNFCT and as input variables for PUTFTOCF. Motion synchronous access is possible here. Example: PUTFTOCF(1, $AA_OVR, 2, 1, 2) The variable $AA_OVR is not allowed here.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. For the functions SYNFCT and PUTFTOCF only certain variables are allowed, for example $AC_DTGPW.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12585 **[Channel %1:] Block %2 variable %3 cannot be changed synchronously with motion**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source symbol
Definitions:	When assigning SYNFCT in motion synchronous actions and result variables, only special variables are allowed. Real-time synchronous access is allowed here. Example: WHEN $AA_IM[AX1]>= 100 DO $AC_TIME=1000. The variable $AC_TIME (time from beginning of block) cannot be written
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. Only certain variables are allowed for the function SYNFCT where real-time synchronous access is possible.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12586 **[Channel %1:] Block %2 motion synchronous action: type conflict in variable %3**

Parameters:	%1 = Channel number %2 = Block number %3 = Source symbol
Definitions:	Type conversion is not possible for online variables $A.. or $V.., which are evaluated or written in the interpolation cycle. Only variables of the same type can be linked or assigned to one another. Example 1: WHENEVER $AA_IM[X] > $A_IN[1] DO ... An online variable of the REAL type (actual value) cannot be compared with a variable of the BOOL type (digital input) The operation is possible if the following change is made: WHENEVER $AA_IM[X] > $A_INA[1] DO ... Example 2: WHENEVER ... DO $AC_MARKER[1]=$AA_IM[X]-$AA_MM[X] Improvement: WHENEVER ... DO $AC_PARAM[1]=$AA_IM[X]-$AA_MM[X]
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program: Use variables of the same type.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12587 **[Channel %1:] Block %2 motion synchronous action: operation/function %3 not allowed**

Parameters:	%1 = Channel number %2 = Block number %3 = Operator/function
Definitions:	The specified function / operator is not permissible for logic operations of real-time variables in motion synchronous actions. The following operators/functions are permissible: - == >= <= > < <> + - * / - DIV MOD - AND OR XOR NOT - B_AND B_OR B_XOR B_NOT - SIN COS TAN ATAN2 SQRT POT TRUNC ROUND ABS EXP LNX SPI
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12588 [Channel %1:] Block %2 motion synchronous action: address %3 not allowed

Parameters:	%1 = Channel number %2 = Block number %3 = Address
Definitions:	- The specified address cannot be programmed in motion-synchronous actions. Example: ID = 1 WHENEVER $A_IN[1]==1 DO D3 - The cutting edge cannot be changed out of a motion-synchronous action.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12589 [Channel %1:] Block %2 motion synchronous action: variable %3 not allowed with modal ID

Parameters:	%1 = Channel number %2 = Block number %3 = Variable name
Definitions:	The modal ID in motion synchronous action must not be formed by means of an on-line variable. Examples: ID=$AC_MARKER[1] WHEN $a_in[1] == 1 DO $AC_MARKER[1] = $AC_MARKER[1]+1 This can be corrected in the following way: R10 = $AC_MARKER[1] ID=R10 WHEN $a_in[1] == 1 DO $AC_MARKER[1] = $AC_MARKER[1]+1 The ID in a synchronous action is always permanent, and cannot be changed in the interpolation cycle.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program: Replace the on-line variable by an arithmetic variable.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The number of global user data blocks is defined in MD18118 $MN_MM_NUM_GUD_MODULES. Here, _N_SGUD_DEF corresponds to block 1, _N_MGUD_DEF corresponds to block 2, _N_UGUD_DEF corresponds to block 3, _N_GUD4_DEF corresponds to block 4 etc. In the directory _N_DEF_DIR there is a file with definitions for global user data, the block number of which is greater than the number of blocks given in the MD. The alarm may, however, also be caused by value zero in one of MD18120 $MN_MM_NUM_GUD_NAMES_NCK, MD18130 $MN_MM_NUM_GUD_NAMES_CHAN and by the definition of a variable with NCK or CHAN in one of the GUD definition files.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Increase MD18118 $MN_MM_NUM_GUD_MODULES. Or, if it already has the correct value, check whether MD18120 $MN_MM_NUM_GUD_NAMES_NCK (if a variable has been defined with attribute NCK) or MD18130 $MN_MM_NUM_GUD_NAMES_CHAN (if a variable has been defined with attribute CHAN) is not zero.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12600 **[Channel %1:] Block %2 invalid line checksum**

Parameters:	%1 = Channel number %2 = Block number
Definitions:	On processing an INI file or when executing a TEA file, an invalid line checksum has been detected.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Correct INI file or correct MD and create new INI file (via "upload").
Program Continuation:	Switch control OFF - ON.

12610 **[Channel %1:] Block %2 accessing single character with call-by-reference parameter not possible %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	An attempt has been made to use a single character access for a call-by-reference parameter.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Temporarily store single characters in user-defined CHAR variable and transfer this.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12620 **[Channel %1:] Block %2 accessing this variable as single character not possible %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	The variable is not a user-defined variable. The single character access is only allowed for user-defined variables (LUD/GUD).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Temporarily store variable in user-defined STRING, process this and put back into storage.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12630 **[Channel %1:] Block %2 skip ID/label in control structure not allowed**

Parameters:	%1 = Channel number %2 = Block number
Definitions:	Blocks with control structures (FOR, ENDIF, etc.) cannot be concealed and must not contain any labels.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program: Reproduce skip ID via an IF query. Write the label alone in the block before the control structure block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12640 **[Channel %1:] Block %2 invalid nesting of control structures**

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

Definitions:	Error in program run: Opened control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) are not terminated or there is no beginning of loop for the programmed end of loop. Example: LOOP ENDIF ENDLOOP
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Correct part program in such a way that all opened control structures are also terminated.
Program Continuation:	Clear alarm with the RESET key. Restart part program

12641 [Channel %1:] Block %2 maximum nesting depth of control structures exceeded

Parameters:	%1 = Channel number %2 = Block number
Definitions:	Max. nesting depth control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) exceeded. At the present time, the max. nesting depth is 8.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Correct part program. If necessary, move parts to a subroutine.
Program Continuation:	Clear alarm with the RESET key. Restart part program

12660 [Channel %1:] Block %2 motion synchronous action: variable %3 reserved formotion synchronous actions and technology cycles

Parameters:	%1 = Channel number %2 = Block number %3 = Variable name
Definitions:	The displayed variable may only be used in motion synchronous actions or in technology cycles. For example, '$R1' may only be used in motion synchronous actions. In standard part programs R parameters are programmed with R1.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12700 [Channel %1:] Block %2 contour definition programming not allowed as modal sub-programis active

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the external language mode, a block is programmed with contour definition and a modal cycle is active at the same time. Because of unclear address assignment (e.g. R = radius for contour definition or return plane for drilling cycle) contour definition programming must not be used when a modal cycle is active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12701	**[Channel %1:] Block %2 illegal interpolation type for contour definition active**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In one contour definition block, G01 is not active as interpolation function. In one contour definition block, the linear interpolation always has to be selected with G01. G00, G02, G03, G33 etc. are not permitted.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. Program linear interpolation G01.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12710	**[Channel %1:] Block %2 illegal language element in external language mode**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The programmed language element is not allowed or unknown in external language mode. Only the language elements from Siemens mode which are used for subprogram calls (except for Lxx) and the language constructs for program repetition with REPEAT (UNTIL) are allowed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. Check that the language command is available in Siemens mode. Switch to Siemens mode with G290. Program the command in the next block and switch back to the external language mode in the following block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12720	**[Channel %1:] Block %2 program number for macro call (G65/G66) missing**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	During macro call with G65/G66 no program number was defined. The program number must be programmed with address "P".
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12722	**[Channel %1:] Block %2 multiple ISO_2/3 macro or cycle calls in the block**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A mixture of cycle and macro calls are programmed in a block, e.g. cycle calls with G81 - G89 together with an M macro in the block or a G65/G66 macro call together with M macros in the block. G05, G08, G22, G23, G27, G28, G29, G30, G50.1, G51.1, G72.1, G72.2 functions (ISO mode) also execute subroutine calls. Only one macro or cycle call can appear in an NC block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Deactivate modal cycles or modal macro calls if one of the above mentioned G functions has been programmed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When programming G07.1 (cylinder interpolation TRACYL), no cylinder radius has been programmed. Selection of the cylinder interpolation (TRACYL) with G07.1 C <cylinder radius> deselect with G07.1 C0. For "C" the name of the rotary axis defined in the TRACYL machine data has to be programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	G07.1 block, program the cylinder radius under the name of the rotary axis for the cylinder interpolation.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12726 [Channel %1:] Block %2 illegal plane selection with parallel axes

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a block with plane selection (G17 _ G19), a basic axis of the coordinate system must not be programmed together with the parallel axis assigned to it.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	For plane selection with G17, G18, G19 either program the basic axis of the coordinate system or the assigned parallel axis.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12728 [Channel %1:] Block %2 distance for double turret not set

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The tool clearance for the double turret head in the SD42162 $SC_EXTERN_DOUBLE_TURRET_DIST is 0.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Enter tool clearance for the double turret head in the SD42162 $SC_EXTERN_DOUBLE_TURRET_DIST.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The machine data MD24100 $MC_TRAFO_TYPE_1, MD24110 $MC_TRAFO_AXES_IN_1[1], MD24210 $MC_TRAFO_AXES_IN_2[1] are incorrectly set for G07.1, G12.1.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Enter valid transformation identifier for TRACYL in MD24100 $MC_TRAFO_TYPE_1 and the rotary axis number in MD24110 $MC_TRAFO_AXES_IN_1[1] or MD24210 $MC_TRAFO_AXES_IN_2[1].
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12740 [Channel %1:] Block %2 modal macro call %3 not possible

Parameters:	%1 = Channel number %2 = Block number, label %3 = Source string
Definitions:	When calling a modal macro no other modal macro, modal cycle or modal subroutine may be active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12750 [channel %1:] block %2 T splitting not possible

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	ISO mode turning: T programming is not possible since the T word cannot be cleary separated by tool number and correction number. Splitting the T word is determined by the machine dates 10888 $MN_EXTERN_DIGITS_TOOL_NO and 10889 $MN_EXTERN_DIGITS_OFFSET_NO. However, only one of the two functions may be active, and at least one function must be active. Alarm occurs only then when no function is active (both MDs = 0) or both functions are active (both MDs <> 0).
Reaction:	Interpreter stop Interface signals are set. Alarm display.
Remedy:	Machine data Adjust 10888 EXTERN_DIGITS_TOOL_NO or 10889 EXTERN_DIGITS_OFFSET_NO. At least one function must be active, but both functions must not be active either.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

12755 [channel %1:] block %2 formatting %3 not possible

Parameters:	%1 = Channel number %2 = Block number, label %3 = Faulty format instructions
Definitions:	The format instructions programmed with the ISOPRINT command are incorrect: - various format instructions %m.nP and %.nP were applied - other format instructions than %P were applied
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- correct ISOPRINT command - within an ISOPRINT command, only format instructions of the same type %m.nP or %.nP may be applied
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14000 [Channel %1:] Block %2 illegal end of file

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Alarm 14000 is output in the following situations: - Parts program was not terminated with M30, M02 or M17. - Executing from external: Download was aborted (e.g. because HMI was switched off).
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set.

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

---

14001   [Channel %1:] Block %2 illegal end of block

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	After system-internal data manipulation (e.g. when reloading from an external source) a part file can end without having LF as the last character.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Read out the part program, modify it with a text editor (e.g., insert blanks or comments before the displayed block), so that after reading it in again the part program has a different structure in the memory.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14005   [Channel %1:] Block %2 program %3 program-specific start disable has been set

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Program name
Definitions:	Program %3 cannot be executed, as the program-specific start disable has been set for this file.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Reset the program-specific start disable for file %3.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14006   [Channel %1:] Block %2 invalid program name %3

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Program name
Definitions:	When selecting or calling an NC program it was found that the program name did not follow NC conventions:
	- The length of the program name, without prefix _N_ and Suffix _MPF / _SPF, must not exceed 24 characters, as otherwise the program name is truncated in the OPI variables.
Reaction:	Alarm display.
Remedy:	- Shorten the name of the program.
	- Suppress the alarm with MD 11415 $MN_SUPPRESS_ALARM_MASK_2 bit 9.
Program Continuation:	Clear alarm with the Delete key or NC START.

14008   [Channel %1:] Block %2 WRITE command writes in the temporary memory area in / _N_EXT_DIR

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	A workpiece is executed from an external data register (Execute from external drives function). The part programs are temporarily stored in the NCK directory /_N_EXT_DIR. An attempt is now made to write in this temporary directory with a WRITE command.

	The alarm is intended to indicate that this data is not stored in the original directory on the external data carrier, and will be lost at the next part program selection because the programs in the directory /_N_EXT_DIR will then be deleted.
Reaction:	Alarm display.
Remedy:	State a directory that remains permanently loaded in the NCK as the target for the WRITE command (e.g. MPF_DIR). The alarm can be suppressed with MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 8.
Program Continuation:	Clear alarm with the Delete key or NC START.

14009   [Channel %1:] Block %2 illegal program path %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Program path
Definitions:	The part program command CALLPATH was called with a parameter (program path) referring to a directory which does not exist in the file system of the NCK.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Modify the CALLPATH instruction such that the parameter contains the complete path name of the loaded directory. - Load the programmed directory in the file system of the NCK.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14010   [Channel %1:] Block %2 invalid default parameter in subroutine call

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a subroutine call with parameter transfer, parameters have been omitted that cannot be replaced by default parameters (call-by-reference parameters or parameters of type AXIS. The other missing parameters are defaulted with the value 0 or with the unit frame in the case of frames).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	The missing parameters must be provided with values in the subroutine call.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14011   [Channel %1:] Block %2 program %3 not existing or will be edited

Parameters:	%1 = Channel number %2 = Block number, label %3 = Program name
Definitions:	A subroutine call was aborted because the called subroutine could not be opened. The subroutine call can be executed via - subroutine designator - CALL / PCALL / MCALL command - SETINT command - M/T function replacement - event-driven program calls (PROG_EVENT) - selection of a PLC ASUB via PI "_N_ASUP__" and/or FB-4 - calling a PLC ASUB via interrupt interface (FC-9) There are various reasons for the alarm: - the subroutine is not in the parts program memory the subroutine - the subroutine is not in the search path (selected directory, _N_SPF_DIR or cycle directories _N_CUS_DIR, _N_CMA_DIR, _N_CST_DIR - the subroutine has not been released or is being edited - faulty absolute path name in subroutine call:

Examples of complete path names: /_N_directoryName_DIR/_N_programmName_SPF or /_N_WKS_DIR/ _N_wpdName_WPD/_N_programmName_SPF. directoryName: MPF, SPF, CUS, CMA, CST (predefined directories). wpdName: application-specific designator for workpiece directories (max. of 24 signs). programmName: Name of subroutine (max. of 24 signs)

- A reload buffer for executing from external was called as subroutine.

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

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Ensure that the subroutine (alarm parameter %3) - is available in the parts program memory - has been released and is not being edited - is available in the search path if not being called via an absolute path name.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The maximum nesting depth of 8 program levels has been exceeded. Subroutines can be called from the main program, and these in turn may have a nesting depth of 7. In interrupt routines the maximum number of levels is 4!
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify the machining program so that the nesting depth is reduced, e.g. using the editor copy a subroutine of the next nesting level into the calling program and remove the call for this subroutine. This reduces the nesting depth by one program level.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14013 [Channel %1:] Block %2 number of subroutine passes invalid

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a subroutine call the programmed number of passes P is zero or negative.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program number of passes between 1 and 9 999.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14014 [Channel %1:] Selected program %3 not available or will be edited

Parameters:	%1 = Channel number %2 = Block number, label %3 = Program name
Definitions:	The selected parts program is not in the NCK memory or the access authorization for the program selection is from a higher level than the current control status. During creation, this program received the protection level of the NC control which was active at the time. In SW 5 or higher a program edited on HMI can no longer be started with NC Start. The alarm will also be issued, if a file other than the specified definition file has been selected for the GUD or macro definition.
Reaction:	Alarm display.
Remedy:	Reload the program in the NCK memory or check and correct the name of the directory (workpiece overview) and the program (program overview) and reselect.

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

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Program name
Definitions:	The execution right currently set in the control (e.g. key switch position 0) is inadequate to execute part program %3.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Raise the execution right to match the protection level of part program %3 - Assign a lower protection level to part program %3 or release (key switch protection level 0)
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The following conflict was detected in a subprogram call per M or T function: In the block referenced by parameter %2: - An M or T function replacement has already been activated - A modal subprogram call is active - A subprogram return jump is programmed - An end of program is programmed - An M98 subprogram call is active (only in external language mode) - T function replacement by D function programming in the same part program line is not possible with active TLC (G43/G44) in ISO2 system.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	An M or T function replacement is only possible if a subprogram call or return jump has not already been performed as a result of other program constructs. The part program must be corrected accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When calling M code subroutine with parameter transfer, an illegal syntax was detected: - Address extension not programmed as a constant. - M function value not programmed as a constant. Note: If a parameter transfer has been programmed via MD10718 $MN_M_NO_FCT_CYCLE_PAR for an M function replacement, the following restriction applies to this M function: both the address extension and the M function value must be programmed for replacement as constants.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Change the programming of the M function.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14018   **[Channel %1:] Block %2 parts program command %3 not executable (protection level setpoint value / actual value: %4)**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Programmed command %4 = Protection level of the command / current protection level
Definitions:	To parts program command %3, a protection level has been assigned that is logically higher (smaller in value) than the current access right, or the command does not exist in the current control configuration.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify parts program. Please see the Siemens Progamming Guide or OEM documentation for the language commands permissible for the relevant system configuration.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14019   **[Channel %1:] block %2 motion synchronous action: %3 wrong value or wrong number of parameters on function or procedure call**

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	- An illegal parameter value was specified in a function or procedure call. - An illegal number of actual parameters was programmed in a function or procedure call.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14020   **[Channel %1:] Block %2 wrong value or wrong number of parameters on function or procedure call**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	- An illegal parameter value was specified in a function or procedure call. - An illegal number of actual parameters was programmed in a function or procedure call.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	- An illegal parameter value was specified in a function or procedure call. - An illegal number of actual parameters was programmed in a function or procedure call.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

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

---

14022 [Channel %1:] Block %2 error on function or procedure call, error code %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Error code
Definitions:	An error occurred during a function or procedure call. The cause of the error is indicated more closely by an error code. The meaning of the error code can be found in the documentation of the function or procedure that caused the error.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14025 [Channel %1:] Block %2 motion synchronous action: illegal modal ID

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In modal motion synchronous actions an illegal ID number has been assigned.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14026 [Channel %1:] Block %2 motion synchronous action: invalid polynomial number in the FCTDEF command

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An FCTDEF command was programmed with a polynomial number that exceeds the maximum value set in MD28252 $MC_MM_NUM_FCTDEF_ELEMENTS.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14027 [Channel %1:] Block %2 motion-synchronous action: Too many technology cycles programmed.

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	You can call a maximum of eight technology cycles with one motion-synchronous action. You exceeded the upper limit.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.

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

14028	**[Channel %1:] Block %2 motion-synchronous action: Technology cycle programmed with too many parameters**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Maximum number of transfer parameters for one technology cycle exceeded.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Change technology cycle
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14030	**[Channel %1:] Block %2 combine OSCILL and POSP during oscillation with infeedmotion**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When oscillating controlled by synchronized actions, the assignment of oscillating and infeed axis (OSCILL) as well as the definition of the infeed (POSP) must be carried out in one NC block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14033	**[Channel %1:] Block %2 involute: no end point programmed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	No end point was programmed for the involute. This is either possible via direct programming with the geometry axis identifiers or by specifying the angle between start and end vector.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14034	**[Channel %1:] Block %2 involute: angle of rotation too large**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	With programming of the angle of rotation (with AR) for involute interpolation, the maximum programmable angle of rotation is limited if the involute is moving towards the basic circle. The maximum value is reached if the involute touches the basic circle. With MD21016 $MC_INVOLUTE_AUTO_ANGLE_LIMIT = TRUE, each angle is accepted without an alarm; if necessary, the angle is automatically limited during interpolation.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14035   **[Channel %1:] Block %2 involute: start point invalid**

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	With involute interpolation, the start point of the involute must be outside the basic circle. The programmed center point or radius must be adapted accordingly.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14036   **[Channel %1:] Block %2 involute: end point invalid**

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	With involute interpolation, the end point of the involute must be outside the basic circle. The programmed center point / radius or end point must be adapted accordingly.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14037   **[Channel %1:] Block %2 involute: radius invalid**

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	With involute interpolation, the programmed radius of the basic circle must be greater than zero.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14038   **[Channel %1:] Block %2 involute not definable: end point error**

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The programmed end point does not lie on the involute defined by the start point, radius and center point of the basic circle. The deviation of the effective end radius from the programmed value is greater than the permissible value specified in MD21015 $MC_INVOLUTE_RADIUS_DELTA.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14039   **[Channel %1:] Block %2 involute: end point programmed several times**

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	With involute interpolation, either the end point with the geometry axis identifiers or the angle of rotation with AR=value can be programmed. Simultaneous programming of end point and angle of rotation in one block is not allowed, since the end point can thus not be defined exactly.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In circular interpolation, either the circle radii for the initial point and the end point are further apart, or the circle center points are further apart, than specified in the machine data. 1. In circle radius programming the starting and end points are identical, thus the circle position is not determined by starting and end points. 2. Radii: The NCK calculates from the present start point and the other programmed circle parameters the radii for the start and the end point. An alarm message is issued if the difference between the circle radii is either - greater than the value in the MD21000 $MC_CIRCLE_ERROR_CONST (for small radii, if the programmed radius is smaller than the quotient of the machine data MD21000 $MC_CIRCLE_ERROR_CONST divided by MD21010 $MC_CIRCLE_ERROR_FACTOR), or - greater than the programmed radius multiplied by the MD21000 $MC_CIRCLE_ERROR_FACTOR (for large radii, if the programmed radius is greater than the quotient of the machine data MD21000 $MC_CIRCLE_ERROR_CONST divided by MD21000 $MC_CIRCLE_ERROR_FACTOR). 3. Center points: A new circle center is calculated using the circle radius at the starting position. It lies on the mid-perpendicular positioned on the connecting straight line from the starting point to the end point of the circle. The angle in the radian measure between both straight lines from the starting point to the center calculated/programmed as such must be lower than the root of 0.001 (corresponding to approx. 1.8 degrees).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check MD21000 $MC_CIRCLE_ERROR_CONST and MD21000 $MC_CIRCLE_ERROR_FACTOR. If the values are within reasonable limits, the circle end point or the circle mid-point of the part program block must be programmed with greater accuracy.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14045 [Channel %1:] Block %2 error in tangential circle programming

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The alarm may have the following causes: The tangent direction is not defined for tangent circle, e.g. because no other travel block has been programmed before the current block. No circle can be formed from start and end point as well as tangent direction because - seen from the start point - the end point is located in the opposite direction to that indicated by the tangent. It is not possible to form a tangent circle since the tangent is located perpendicular to the active plane. In the special case in which the tangent circle changes to a straight line, several complete circular revolutions were programmed with TURN.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14048	**[Channel %1:] Block %2 wrong number of revolutions in circle programming**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An impermissible number of full revolutions was specified during circle programming. The number of revolutions may neither be negative nor exceed 1000000.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

14050	**[Channel %1:] Block %2 nesting depth for arithmetic operations exceeded**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	For calculating arithmetic expressions in NC blocks, an operand stack with a fixed set size is used. With very complex expressions, this stack can overflow. This may also occur with extensive expressions in synchronized actions.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Divide up complex arithmetic expressions into several simpler arithmetic blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14051	**[Channel %1:] Block %2 arithmetic error in part program**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	- In calculating an arithmetic expression, an overflow has occurred (e.g. division by zero) - In a data type, the representable value range has been exceeded
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Analyze the program and correct the defective point in the program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14055	**[Channel %1:] Block %2 impermissible NC language substitution, error code %3**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Error code
Definitions:	This alarm occurs in conjunction with an NC language substitution configured in MD30465 $MA_AXIS_LANG_SUB_MASK. Error code %3 gives more detailed information about the cause of the problem: Error code: 1: Several events had been programmed, causing the replacement cycle to be called. Only one substitution is allowed per part program line. 2: A non-modal synchronized action had also been programmed for the part program line with the NC language substitution. 3: The system variables $P_SUB_SPOSIT and $P_SUB_SPOSMODE were called outside a replacement cycle.
Reaction:	Correction block is reorganized. Interpreter stop Interface signals are set. Alarm display.
Remedy:	Modify the NC program

Diagnostics Manual

158   Diagnostics Manual, 12/2012, 6FC5398-6DP10-0BA0

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

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	With "Differential block skip", a skip level greater than 7 has been specified. (In packet 1 specification of a value for the skip level is rejected by the converter as a syntax error, i.e. the only possibility is a "Suppress block" ON/OFF on one level).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Enter a skip level (number behind the slash) less than 8.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14065 [channel %1:] block %2 error in SPRINT/ISOPRINT command: error code %4 information %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Additional information %4 = Error code
Definitions:	When interpreting the SPRINT or ISOPRINT command, an error was detected which was described in more detail by parameter %4. Parameter %3 may supply additional information on the incurring problem. List of error codes (parameter %4): 1: invalid format description %3 recognized 2: Format %.nP: overrange when converting into 32 bit number 3: Format %P: Character %3 cannot be converted with the code selected by MD 10750 / $MN_SPRINT_FORMAT_P_CODE. 4: Maximum string length of 400 bytes exceeded 5: SPRINT/ISOPRINT command programmed with an invalid number of parameters 6: SPRINT/ISOPRINT parameters programmed with impermissible data type 7: Format %m.nP: overrange due to parameter n with MD 10751 / $MN_SPRINT_FORMAT_P_DECIMAL = 0
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct SPRINT and/or ISOPRINT command(s).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14066 [channel %1:] block %2 error when outputting to external device on command %3, error code: %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Part program command %4 = Error code
Definitions:	When processing the commands ISOOPEN, ISOPRINT, ISOCLOSE or EXTOPEN, WRITE, EXTCLOSE, an error was detected which is described in detail by the error code. List of error codes: 1: external device cannot be opened 2: external device is not configured 3: external device configured with invalid path 4: no access rights for external device 5: external device already exclusively assigned 6: external device already assigned in shared mode 7: file length greater than LOCAL_DRIVE_MAX_FILESIZE 8: maximum number of external devices exceeded 9: option not set for LOCAL_DRIVE

11: V24 already occupied by Easy-Message function
12: Append/Overwrite specification is inconsistent with extdev.ini
14: external device not assigned or opened
15: error when writing to external device
16: invalid external path programmed
21: error when closing external device
22: external device not installed (mounted)
90: Timeout

Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Correcting the parameterization of the ISOCLOSE, ISOPRINT or ISOCLOSE command. See also MD 10830 $MN_EXTERN_PRINT_DEVICE and MD 10831 $MN_EXTERN_PRINT_MODE. Check the configuration of the external device on the CF card in /user/sinumerik/nck/extdev.ini and /oem/sinumerik/nck/extdev.ini. Check the connection and functioning of the external device.
Program Continuation:	Clear alarm with the RESET key. Restart part program

14070 [Channel %1:] Block %2 memory for variables not sufficient for subroutine call

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A called subroutine cannot be processed (opened), either because the internal data memory to be created for general purposes is not large enough, or because the available memory for the local program variables is too small. The alarm can only occur in MDI mode.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Analyze the part program section: 1. Has the most useful data type always been selected in the variable definitions? (For example REAL for data bits is poor; BOOL would be better) 2. Can local variables be replaced by global variables?
Program Continuation:	Clear alarm with the RESET key. Restart part program

14080 [Channel %1:] Block %2 jump destination %3 not found

Parameters:	%1 = Channel number %2 = Block number, label %3 = Jump destination
Definitions:	In conditional and unconditional jumps, the jump destination within the program must be a block with a label (symbolic name instead of block number). If no jump destination has been found with the given label when searching in the programmed direction, an alarm is output. For parameterizable returns with RET to block number or label, the jump destination within the program must be a block with the block number or label (symbolic name instead of block number). For returns over several levels (parameter 2), the jump destination must be a block within the program level you jumped to. For returns with a string as return destination, the search string must be a name known in the control and the search string must be preceded in the block by a block number and/or a label only.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check NC part program for the following possible errors: 1. Check whether the target designation is identical with the label. 2. Is the jump direction correct? 3. Has the label been terminated with a colon?
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Start or end label
Definitions:	The start point for repetition of the program part with CALL <program name> BLOCK <start label> TO <end label> has not been found or the same program part repetition has been called recursively.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check the start and end labels for programming repetition in the user program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The instruction 'TML()' may only be used in the subprogram, which replaces the T command.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	An axis position larger than 3.40e+38 increments has been programmed. This alarm can be suppressed with bit11 in MD11410 $MN_SUPPRESS_ALARM_MASK.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Index
Definitions:	A function has been programmed or triggered which is not permitted in the current program context. The function in question is encoded in the "Index" parameter: Index == 1: "RET" command was programmed in the main program level Index == 2: Conflict between "Cancel level"/"Clear number of passes" and "Implicit GET" Index == 3: Conflict ASUB start immediately after selection of overstore (up to P3) Index == 4: MD10760 $MN_G53_TOOLCORR = 1 : SUPA/G153/G53 programmed in G75 Index == 5: POSRANGE command not programmed in synchronized action Index == 6: SIRELAY command not programmed in synchronized action Index == 7: GOTOF/GOTOB/GOTO command programmed with string variable in synchronized action Index == 8: COA application "Cutting generator" not active Index == 9: Tool radius compensation active in G75 Index == 10: Number of return levels too big, with RET (...xy) across several program levels Index == 11: The function is not implemented for this variable

Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Index == 1: Substitute "RET" command with M17/M30 Index == 2: Insert an auxiliary block (e.g. M99) after the subroutine call to which the "Cancel level"/"Clear number of passes" refers Index == 3: Overstore an auxiliary block (e.g. M99), then start ASUB (up to P3) Index == 4: With MD10760 $MN_G53_TOOLCORR = 1: Do not activate SUPA/G53/G153 in the G75 block Index == 5: Program POSRANGE command in synchronized action Index == 6: Program SIRELAY command in synchronized action Index == 7: Program GOTOF/GOTOB/GOTO command with block number or label Index == 8: Load COA application "Cutting generator" Index == 9: Tool radius compensation active in G75
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	One of the following programming errors has occurred: 1. The keyword WAITP(x) "Wait with block change until the specified positioning axis has reached its end point" has been used for an axis that is not a positioning axis. 2. G74 "Reference point approach from the program" has been programmed for a spindle. (Only axis addresses are permitted). 3. The keyword POS/POSA has been used for a spindle. (The keywords SPOS and SPOSA must be programmed for spindle positioning). 4. If the alarm occurs with the "Rigid tapping" function (G331), the following causes are conceivable: - The master spindle is not in position-controlled mode. - Incorrect master spindle - Master spindle without encoder 5. An axis name was programmed which no longer exists (e.g. when using axial variables as an index). Or NO_AXIS has been programmed as an index. 6. If 14092 is output as a note with alarm 20140 Motion-synchronous action: traversing of command axis, the following causes are possible: - The axis is currently being traversed by the NC program already. - An overlaid movement is active for the axis. - The axis is active as following axis of a coupling. - An interpolation compensation such as a temperature compensation is active for the axis.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Correct the part program according to which of the above errors is involved. - Program SPOS. - Set the correct master spindle with SETMS.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The radius entered for radius programming is too small, i.e. the programmed radius is smaller than half of the distance between start and end point.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.

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

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	During the program run, a variable value assignment or an arithmetic operation has caused data to be processed in such a way that they have to be converted to another type. This would lead to the value range being exceeded.
	Value ranges of the various variable types:
	- REAL: Property: Fractional number with dec. pt., value range: +/-(2-1022-2+1023)
	- INT: Property: Integers with signs, value range: +/-(231-1)
	- BOOL: Property: Truth value TRUE, FALSE, value range: 0,1
	- CHAR: Property: 1 ASCII character, value range: 0-255
	- STRING: Property: Character string (max. 100 values), value range: 0-255
	- AXIS: Property: Axis addresses, value range: Axis names only
	- FRAME: Property: Geometric information, value range: As for axis paths
	Overview of type conversions:
	- from REAL to: REAL: yes, INT: yes*, BOOL: yes1), CHAR: yes*, STRING: -, AXIS: -, FRAME: -
	- from INT to: REAL: yes, INT: yes, BOOL: yes1), CHAR: if value 0 ...255, STRING: -, AXIS: -, FRAME: -
	- from BOOL to: REAL: yes, INT: yes, BOOL: yes, CHAR: yes, STRING: -, AXIS: -, FRAME: -
	- from CHAR to: REAL: yes, INT: yes, BOOL: yes1), CHAR: yes, STRING: yes, AXIS: -, FRAME: -
	- from STRING to: REAL: -, INT: -, BOOL: yes2), CHAR: only if 1 character, STRING: yes, AXIS: -, FRAME: -
	- from AXIS to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: yes, FRAME: -
	- from FRAME to: REAL: -, INT: -, BOOL: -, CHAR: -, STRING: -, AXIS: -, FRAME: yes
	1) Value <> 0 corresponds to TRUE, value ==0 corresponds to FALSE.
	2) String length 0 => FALSE, otherwise TRUE.
	3) If only 1 character.
	It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Modify the program section such that the value range is not exceeded, e.g. by a modified variable definition.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The called function AXNAME - conversion of the transferred parameters of the STRING type to an axis name (return value) of the AXIS type - has not found this axis identifier in the machine data.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check the transferred parameter (axis name) of the function AXNAME to determine whether a geometry, channel or machine axis of this name has been configured by means of the machine data:
	MD10000 $MN_AXCONF_MACHAX_NAME_TAB
	MD20060 $MC_AXCONF_GEOAX_NAME_TAB
	MD20080 $MC_AXCONF_CHANAX_NAME_TAB
	Select the transfer string in accordance with the axis name, and change the axis name in the machine data if necessary. (If a change of name is to take place via the NC part program, this change must first be validated by means of a "POWER-ON").
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The string is not a valid INT or REAL number.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program. If it is an entry, then you can check whether the string is a number via the preset function ISNUMBER (with the same parameter).
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The result of string chaining returns a string which is greater than the maximum string length laid down by the system.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Adapt part program. With the function STRLEN, it is also possible to query the size of the sum string before executing the chaining operation.
Program Continuation:	Clear alarm with the RESET key. Restart part program

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	On assigning an array by means of SET, more initialization values than existing array elements have been specified in the program run.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Reduce the number of initialization values.
Program Continuation:	Clear alarm with the RESET key. Restart part program

14140   **[Channel %1:] Block %2 position programming without transformation not allowed**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Position information was programmed for an axis position but no transformation was active.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify the program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

14144   **[Channel %1:] Block %2 PTP movement not allowed**

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

Definitions:	The PTP G code was programmed for a movement other than G0 or G1.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Modify the program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14146   [Channel %1:] Block %2 CP or PTP movement without transformation not allowed

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The CP or PTP G code was programmed for a movement but no transformation was active.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Modify the program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14157   [Channel %1:] Block %2 illegal interpolation type with MOVT

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Linear or spline interpolation must be active with MOVT (G0, G1, ASPLINE, BSPLINE, CSPLINE).
Reaction:	Correction block is reorganized.
	Interpreter stop
	Interface signals are set.
	Alarm display.
Remedy:	Modify program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14159   [Channel %1:] Block %2 more than two angles programmed with ROTS or AROTS

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Frame rotations are described using space angles with the language commands ROTS or AROTS. A maximum of two angles can be programmed.
Reaction:	Correction block is reorganized.
	Interpreter stop
	Interface signals are set.
	Alarm display.
Remedy:	Modify program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	If variant C (tool length acts on the programmed axis) is activated by machine data MD20380 $MC_TOOL_CORR_MODE_G43G44 for tool length compensation with H word and G43/G44 in ISO_2 mode, at least one geometry axis must be specified.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.

	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Change MD20380 $MC_TOOL_CORR_MODE_G43G44 or the part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = ISO H/D number
	%4 = Tool number
Definitions:	When an H or D number is programmed in ISO_2 or ISO_3 mode, it must be available in the active tool. The active tool may also be the last tool loaded on the master spindle or master toolholder. This alarm is output if there is no H or D number on this tool.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Set ISO H/D number correctly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14166 [Channel %1:] Block %2 error %3 when programming a tool length offset with TOFF / TOFFL

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Error code
Definitions:	An error occurred while programming a tool length offset with TOFF or TOFFL. More information about the type of error is given by the error code number:
	Error code
	1 At least one tool length offset component has been programmed twice in one block (with TOFF).
	2 At least one tool length offset component has been programmed twice in one block (with TOFFL).
	3 Tool length offset components have been programmed in one block with both TOFF and TOFFL.
	4 An index must be declared when a tool length offset is programmed with TOFF, the form TOFF=.... is not permissible.
	5 An illegal index was declared when programming TOFFL (permissible values 1..3).
	6 An illegal axis was declared as the index when programming TOFF. Only geometry axes are permitted.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct errors in program block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	If tool compensation (G43/G44) is activated in language mode ISO_M, the linear type of interpolation must be active.
Reaction:	Correction block is reorganized.
	Local alarm reaction.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14180   **[Channel %1:] Block %2 H number %3 is not defined**

Parameters:	%1 = Channel number %2 = Block number, label %3 = H number of ISO mode
Definitions:	The specified H number is not assigned to a tool (ISO_M).
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14185   **[Channel %1:] Block %2 D number %3 is not defined**

Parameters:	%1 = Channel number %2 = Block number, label %3 = D number of ISO mode
Definitions:	The specified D number is not assigned to a tool (language mode ISO_M).
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14197   **[Channel %1:] Block %2 D number and H number programmed simultaneously**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A D word and H word have been programmed simultaneously.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If an offset is active in the tool direction, block change is not possible if this would change the assignment of the offset axes to the channel axes (plane change, tool change, cutter <=> turning tool, geometry axis exchange).
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	- Modify part program. - Reduce the offset in tool direction to zero.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14199   **[Channel %1:] Block %2 illegal plane change for tool with diameter component**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If a tool has a wear or length component which is evaluated as a diameter for the facing axis (bit 0 and/or bit 1 in MD20360 $MC_TOOL_PARAMETER_DEF_MASK is set) and bit 2 of this MD is also set, this tool may only be used in the plane active on tool selection. A plane change results in an alarm.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program. Reset bit 2 in MD20360 $MC_TOOL_PARAMETER_DEF_MASK.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

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

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the endpoint specification of a traversing block with G00, G01, G02 or G03 in polar coordinates, the polar radius entered for the keyword RP=... is negative. Definition of terms: - Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03). - New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct NC part program - permissible inputs for the pole radius are only positive absolute values that specify the distance between the current pole and the block end point. (The direction is defined by the polar angle AP=...).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14210   **[Channel %1:] Block %2 polar angle too large**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In specifying the endpoints in a traversing block with G00, G01, G02 or G03 in polar coordinates, the value range of the polar angle programmed under the keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees. Definition of terms: - Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03). - New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... referred to the last programmed point in the plane, G111 ... referred to the zero point of the current workpiece coordinate system (Work), G112 ... referred to the last pole.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14250 [Channel %1:] Block %2 negative pole radius

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

Definitions:
In redefining the pole with G110, G111 or G112 in polar coordinates, the pole radius specified under keyword RP=... is negative. Only positive absolute values are permitted.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

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

Remedy:
Correct the NC part program. Permissible inputs for the pole radius are only positive, absolute values that specify the distance between the reference point and the new pole. (The direction is defined with the pole angle AP=...).

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

14260 [Channel %1:] Block %2 pole angle too large

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

Definitions:
In redefining the pole with G110, G111 or G112 in polar coordinates, the value range of the pole angle specified under keyword AP=... has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.
Definition of terms:
- Specification of end of block point with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
- New definition of the pole with polar angle and pole radius, referring to the reference point selected with the G function. G110 ... last programmed point in the plane, G111 ... zero point of the current work, G112 ... last pole

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

Remedy:
Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.

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

14270 [Channel %1:] Block %2 pole programmed incorrectly

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

Definitions:
When defining the pole, an axis was programmed that does not belong to the selected processing level. Programming in polar coordinates always refers to the plane activated with G17 to G19. This also applies to the definition of a new pole with G110, G111 or G112.

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

Remedy:
Correct the NC part program. Only the two geometry axes may be programmed that establish the current machining plane.

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

14280 [Channel %1:] Block %2 polar coordinates programmed incorrectly

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

Definitions:
The end point of the displayed block has been programmed both in the polar coordinate system (with AP=..., RP=...) and in the Cartesian coordinate system (axis addresses X, Y,...).

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct the NC part program - the axis motion may be specified in one coordinate system only.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14300 [Channel %1:] Block %2 overlaid handwheel motion activated incorrectly

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Handwheel override has been called up incorrectly: - 1st For positioning axes: - Handwheel override programmed for indexing axes, - No position programmed, - FA and FDA programmed for the same axis in the block. - 2nd For contouring axes: - No position programmed, - G60 not active, - 1st G group incorrect (only G01 to CIP).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14320 [Channel %3:] Axis %4: handwheel %1 used twice (%2)

Parameters:	%1 = Handwheel number %2 = Use %3 = Channel %4 = Axis
Definitions:	Informational alarm indicating that the mentioned handwheel is used twice: The second parameter provides the explanation: 1: Block with axial handwheel override for this axis cannot be executed as the handwheel for this axis performs a DRF movement 2: Block with velocity override of the path cannot be executed as the handwheel performs a DRF movement for this axis of the path 3: Block with contour handwheel cannot be executed as the handwheel performs a DRF movement for this axis of the path 4: PLC axis with axial handwheel override cannot be started immediately as the handwheel performs a DRF movement for this axis 5: The axis is a reciprocating axis with axial handwheel override; the reciprocating movement cannot be started immediately as the handwheel performs a DRF movement for this axis 6: The DRF movement for this axis cannot be executed as an axial handwheel override is active for this axis with the handwheel 7: The DRF movement for this axis cannot be executed as a velocity override of the path with the handwheel is active and the axis belongs to the path 8: The DRF movement for this axis cannot be executed as the contour handwheel is active with this handwheel and the axis belongs to the path 9: The DRF movement for this axis cannot be executed as the axis is a PLC axis with handwheel override that is active with this handwheel 10: The DRF movement for this axis cannot be executed as the axis is active as reciprocating axis with handwheel override with this handwheel
Reaction:	Alarm display.
Remedy:	Use the handwheel for one purpose at a time only.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

14400   **[Channel %1:] Block %2 tool radius compensation active at transformation switchover**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A change of transformation is not allowed when tool radius compensation is active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Perform tool radius compensation in the NC part program with G40 (in a block with G00 or G01) before performing a transformation change.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14411   **[Channel %1:] Block %2 tool radius compensation active at geometry axis changeover**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	It is not permissible to change the assignment of geometry axes to channel axes when tool radius compensation is active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14414   **[Channel %1:] Block %2 GEOAX function: incorrect call**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The parameters for calling the GEOAX(...) are incorrect. Possible causes: - Uneven number of parameters. - More than 6 parameters were specified. - A geometry axis number was programmed which was less than 0 or greater than 3. - A geometry axis number was programmed more than once. - An axis identifier was programmed more than once. - An attempt was made to assign a channel axis to a geometry axis which has the same name as one of the channel axes. - An attempt was made to assign a channel axis to a geometry axis lacking IPO functionality (see MD30460 $MA_BASE_FUNCTION_MASK, Bit8). - An attempt was made to remove a geometry axis with the same name as one of the channel axes from the geometry axis grouping.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program or correction block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14420   **[Channel %1:] Block %2 index axis %3 frame not allowed**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis
Definitions:	The axis is to be traversed as an indexing axis, but a frame is active. This is not allowed by MD32074 $MA_FRAME_OR_CORRPOS_NOTALLOWED.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. Modify part program. Change MD32074 $MA_FRAME_OR_CORRPOS_NOTALLOWED.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14500   [Channel %1:] Block %2 illegal DEF or PROC instruction in the part program

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	NC part programs with high-level language elements are divided into a preceding definition part followed by a program part. The transition is not marked specifically; a definition statement is not allowed to follow the 1st program command.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Put definition and PROFC statements at the beginning of the program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14510   [Channel %1:] Block %2 PROC instruction missing on subroutine call

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In subroutine calls with parameter transfer ("call-by-value" or "call-by-reference") the called subroutine must begin with a PROC statement.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Define the subroutine in accordance with the type used. 1. Conventional subroutine structure (without parameter transfer): % SPF 123456 : M17 2. Subroutine structure with keyword and subroutine name (without parameter transfer): PROC UPNAME : M17 ENDPROC 3. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-value"): PROC UPNAME (VARNAME1, VARNAME2, ...) : M17 ENDPROC 4. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-reference"): PROC UPNAME (Typ1 VARNAME1, Typ2 VARNAME2, ...) : M17 ENDPROC
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14520   [Channel %1:] Block %2 illegal PROC instruction in data definition section

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The PROC statement may only be programmed at the beginning of the subroutine.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify NC part program appropriately.

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

14530 [Channel %1:] Block %2 EXTERN and PROC instruction do not correspond

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Subroutines with parameter transfer must be known before they are called in the program. If the subroutines are always available (fixed cycles) the control establishes the call interfaces at the time of system power-up. Otherwise an EXTERN statement must be programmed in the calling program. Example: N123 EXTERN UPNAME (TYP1, TYP2, TYP3, ...) The type of the variable must match the type given in the definition (PROC statements) or it must be compatible with it. The name can be different.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Check the variable types in the EXTERN and the PROC statements for correspondence and correctness.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14600 [Channel %1:] Block %2 reload buffer %3 cannot be established

Parameters:	%1 = Channel number %2 = Block number, label %3 = File name
Definitions:	The download buffer for "execute from external" could not be created. Possible causes: - Not enough memory available (for minimum see MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE) - No resources available for HMI NCK communication (see MD18362 $MN_MM_EXT_PROG_NUM) - The file already exists
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	- Release memory, e.g. by deleting part programs - Modify MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE and/or MD18362 $MN_MM_EXT_PROG_NUM.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14601 [Channel %1:] Block %2 reload buffer could not be deleted

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The reload buffer for "execute from external" could not be deleted. Possible cause: - HMI/PLC communication was not terminated.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	All reload buffers are cleared on POWER ON.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14602 [Channel %1:] Block %2 timeout while reloading from external.

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

Definitions:	No connection could be made to the HMI while reloading external subprograms (EXTCALL) or executing from external drives) within the monitoring time set in MD10132 $MN_MMC_CMD_TIMEOUT.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	- Check the connection to the HMI - Increase MD10132 $MN_MMC_CMD_TIMEOUT.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14603   [Channel %1:] Block %2 timeout during execution from external source.

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If a program is selected for execution from external source, it will be expected that the first part program line can be read from the reload buffer within 60s after part program start. Otherwise, part program processing will be aborted with alarm 14603 due to the assumption that the connection to the HMI or the external device is faulted.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Check the connection to the HMI and repeat selection of the program that is to be executed from external source.
Program Continuation:	Clear alarm with the RESET key. Restart part program - Acknowledge the alarm by pressing the RESET key - Repeat program selection - Start the part program

---

14610   [Channel %1:] Block %2 compensation block not possible

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An alarm was output which could be eliminated basically via program correction. Since the error occurred in a program which is processed from external, a compensation block/program correction is not possible.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	- Abort program with reset. - Correct program on HMI or PC. - Restart reloading (possibly with block search and interrupt location).
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14615   [Channel %1:] An error occurred while handling the function 'syntax check': identifier %3

Parameters:	%1 = Channel number %2 = Is not used %3 = Error code
Definitions:	An error occurred while handling the function syntax check via the PI services _N_CHKSEL, _N_CHKRUN, _N_CHKABO and _N_SEL_BL. Parameter %3 describes the error situation more closely: Value 1: An invalid line number was transferred with the PI service _N_SEL_BL 2: An invalid line number for the range end was transferred with the PI service _N_CHKRUN 3: PI service _N_CHKSEL was activated although a block selection (PI service _N_SEL_BL) was active for the selected program.
Reaction:	Alarm display.

Remedy:	Value
	1: Supply PI service _N_SEL_BL with the correct line number
	2: Supply PI service _N_CHKRUN with the correct line number for the range end
	3: Ensure that the channel is in reset status before activating the PI service _N_CHKSEL.
Program Continuation:	Clear alarm with the Delete key or NC START.

14650 [Channel %1:] Block %2 SETINT instruction with invalid ASUP input

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Asynchronous subroutines (ASUBs) are subroutines that are executed following a hardware input (interrupt routine started by a rapid NCK input).
	The NCK input number must lie between 1 and 8. It is assigned a priority from 1 to 128 (1 is the highest priority) in the SETINT instruction with the keyword PRIO = ...
	Example:
	If NCK input 5 changes to "1 signal", the subroutine AB-HEB_ Z should be started with the highest priority.
	N100 SETINT (5) PRIO = 1 ABHEB_Z
	Restriction for SW PLC2xx: The number of the NCK input must be 1 or 2.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Program the NCK input of the SETINT statement with a value of not less than 1 or greater than 8.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14660 [Channel %1:] Block %2 SETINT instruction with invalid priority

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The NCK input number must lie between 1 and 8. It is assigned a priority from 0 to 128 (1 is the highest priority) in the SETINT instruction with the keyword PRIO = ...
	Example:
	If NCK input 5 changes to "1-signal" the subroutine ABHEB_ Z should be started with the highest priority.
	N100 SETINT (5) PRIO = 1 ABHEB_Z
	Restriction for SW PLC2xx: The number of the NCK input must be 1 or 2.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Program the priority of the NCK input with a value of not less than 1 or greater than 128.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14700 [Channel %1:] Block %2 timeout during command to interpreter

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	A timeout has occurred in control-internal commands such as ANWAHL (part program selection), RESET (channel reset), REORG (reorganization of the preprocessing buffer) and NEWCONFIG (change in the configuration-specific machine data = restart).
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Please inform the authorized personnel/service department.
	If the runtime error occurred as the result of a temporary excessive load on the system (e.g. in the HMI area or in OEM applications) error-free execution is possible on repeating the program or operator action.
	Otherwise, place a support request with the error text under: http://www.siemens.com/automation/support-request
Program Continuation:	Switch control OFF - ON.

14701   [Channel %1:] Block %2 number of available NC blocks reduced by %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of non-available blocks
Definitions:	After reset, it has been found that the number of available blocks has decreased compared with the last reset. This is due to a system error. Part program execution can be resumed after the alarm has been acknowledged. If the number of blocks no longer available is less than MD28060 $MC_MM_IPO_BUFFER_SIZE, then the POWERON alarm 14700 is output.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Proceed as in the case of a system error.
Program Continuation:	Clear alarm with the RESET key. Restart part program

14710   [Channel %1:] Block %2 error in initialization sequence in function %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Identifier of the function which caused the error
Definitions:	Initialization blocks are generated (or not) after control power-up, (program) RESET and (program) START, depending on the settings in machine data MD20110 $MC_RESET_MODE_MASK and MD20112 $MC_START_MODE_MASK. Errors can occur because of incorrect machine data settings. The errors are output with the same error messages as would appear if the function had been incorrectly programmed in the part program. This alarm is also generated in order to indicate that an error relates to the initialization sequence. Parameter %3 specifies which function triggers the alarm: Control power-up and (program) RESET: Value: 0: Error during synchronization preprocessing/main run 1: Error on selection of tool length compensation 2: Error on selection of transformation 3: Error on selection of work offset The macro definitions and cycle interfaces are also read in during the power-up procedure. If an error occurs here, this is indicated by value = 4, or value = 5 6: Error creating 2 1/2 D protection zones during power up. (Program) START: Value 100: Error during synchronization preprocessing/main run 101: Error on selection of tool length compensation 102: Error on selection of transformation 103: Error on selection of synchronized spindle 104: Error on selection of work offset Particularly when tool management is active, it is possible that a tool on the spindle or the toolholder is disabled but still needs to be activated. These tools are automatically activated on RESET. On START, machine data MD22562 $MC_TOOL_CHANGE_ERROR_MODE can be used to specify whether an alarm is to be generated or an automatic bypass strategy selected. If the parameter contains 3 values from 200 to 203, this means that an insufficient number of NC blocks is available for NC block preparation on certain commands (ASUB start, overstore selection, teach-in). Remedy: Increase machine data MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.
Reaction:	Interpreter stop Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. If parameter %3= 0 -3: If the alarm or alarms occur on RESET: Check the settings of machine data MD20110 $MC_RESET_MODE_MASK, MD20120 $MC_TOOL_RESET_VALUE, MD20121 $MC_TOOL_PRESEL_RESET_VALUE, MD20122 $MC_TOOL_RESET_NAME (only if tool management is active), MD20130 $MC_CUTTING_EDGE_RESET_VALUE, MD20132 $MC_SUMCORR_RESET_VALUE, MD20126 $MC_TOOL_CARRIER_RESET_VALUE, MD20150 $MC_GCODE_RESET_VALUES, MD20154 $MC_EXTERN_GCODE_RESET_VALUES, MD20140 $MC_TRAFO_RESET_VALUE, MD21330 $MC_COUPLE_RESET_MODE_1, MD24002 $MC_CHBFRAME_RESET_MASK If parameter %3= 100 - 104: Check the setting of MD20112 $MC_START_MODE_MASK and the machine data specified under '..._RESET_...'. If tool management is active, if necessary remove the tool stated in the associated alarm from the toolholder/spindle and cancel the 'disabled' status. If parameter %3= 4 or 5: Check macro definitions in _N_DEF_DIR Check cycle directories _N_CST_DIR and _N_CUS_DIR If parameter %3= 6: Alarm 18002 or 18003 was also issued. This alarm contains the number of the incorrectly defined protection zone and an identifier of what is incorrect about the protection zone. The system variables must be appropriately corrected. If Parameter %3= 200 bis 203: Increase MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14750   [Channel %1:] Block %2 too many auxiliary functions programmed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	More than 10 auxiliary functions have been programmed in an NC block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check whether all auxiliary functions are necessary in one block - modal functions need not be repeated. Create separate auxiliary function block or divide the auxiliary functions over several blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14751   [Channel %1:] Block %2 resources for motion synchronous actions not sufficient (code: %3)

Parameters:	%1 = Channel number %2 = Block number, label %3 = Identifier
Definitions:	Processing of motion-synchronous actions requires resources that are configured using MD28060 $MC_MM_IPO_BUFFER_SIZE, MD28070 $MC_MM_NUM_BLOCKS_IN_PREP, MD28251 $MC_MM_NUM_SAFE_SYNC_ELEMENTS, MD28250 $MC_MM_NUM_SYNC_ELEMENTS, and MD28253 $MC_MM_NUM_SYNC_STRINGS. If these resources are insufficient for the execution of the part program, then this alarm is issued. Parameter %3 shows which resource has run out: Increase identifier <= 2: MD28060 $MC_MM_IPO_BUFFER_SIZE or MD28070 $MC_MM_NUM_BLOCKS_IN_PREP. Increase identifier > 2: MD28250 $MC_MM_NUM_SYNC_ELEMENTS, MD28251 $MC_MM_NUM_SAFE_SYNC_ELEMENTS. Increase identifier 7: MD28253 $MC_MM_NUM_SYNC_STRINGS.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct part program or increase resources.

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

14752   [Channel %1:] Block %2 DELDTG | STOPREOF conflict

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In a block of motion synchronous actions referring to a motion block, both DELDTG (delete distance-to-go) and STOPREOF (preprocessing stop) have been programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	The functions DELDTG and STOPREOF exclude each other in a block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14753   [Channel %1:] Block %2 motion synchronous action: %3 illegal interpolation type

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	The active interpolation type (e.g. 5-axis interpolation) is not allowed for the motion synchronous action or for the function "Several feeds".
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14754   [Channel %1:] Block %2 motion synchronous action: %3 wrong feed type

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	The active feed type is not allowed for the motion synchronous action or for the function "Several feeds".
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14756   [Channel %1:] Block %2 motion synchronous action: %3 wrong value

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	Illegal value.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.

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

14757 [Channel %1:] Block %2 motion synchronous action and wrong type

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Programmed combination between action and type of motion synchronous action is not allowed. - RET allowed in technology cycle only - Function "Several feeds" not allowed in technology cycle - H and M function outputs not allowed with WHENEVER, FROM and DO - MEASA / MEAWA / MEAC with WHENEVER, FROM and DO not allowed - DELDTG and STOPREOF allowed only in blockwise synchronous action with WHEN and EVERY
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14758 [Channel %1:] Block %2 programmed value not available

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The synchronous variables $AA_LOAD, $AA_TORQUE, $AA_POWER and $AA_CURR are available only for the SIMODRIVE611D drive. They are activated by the machine data MDC 36730 $MA_DRIVE_SIGNAL_TRACKING. The system variable $VA_IS: Safe Actual Position is available only if the MD36901 $MA_SAFE_FUNCTION_ENABLE has been set and the option $ON_NUM_SAFE_AXES has been set to a sufficient size.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify program or machine data.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14759 [Channel %1:] Block %2 motion synchronous action and wrong axis type

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When there are several feeds, a spark-out time, or a retraction stroke for path motions, at least one GEO axis must be programmed. If the block also contains synchronous axes and there are several feeds, the feedrate for the synchronous axes is matched implicitly. No retraction stroke takes place for synchronous axes. However, after retraction stroke or spark-out time, the distance-to-go is also deleted in the block for the synchronous axes. The alarm is no longer used on P3.2.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program the axis as positioning axis with axial feed, return stroke or spark-out time.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14760 [Channel %1:] Block %2 auxiliary function of a group programmed repeatedly

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The M and H functions can be divided up as required over machine data in groups in any variation. Auxiliary functions are thus put into groups that mutually preclude several individual functions of one group. Within one group only one auxiliary function is advisable and permissible.
Reaction:	Correction block is reorganized.

Interface signals are set.
Alarm display.

| **Remedy:** | Please inform the authorized personnel/service department. Only program one help function per help function group. (For the group division, see the machine manufacturer's programming guide). |
| **Program Continuation:** | Clear alarm with NC START or RESET key and continue the program. |

14761 [Channel %1:] Block %2 motion synchronous action: DELDTG function not allowed with active tool radius compensation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Rapid delete distance-to-go for synchronous actions is not allowed with DELDTG when tool radius compensation is active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Deactivate tool radius compensation before performing rapid delete distance-to-go and then reselect or as of SW 4.3: "Delete distance-to-go without preparation".
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14762 [Channel %1:] Block %2 too many PLC variables programmed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The number of programmed PLC variables has exceeded the maximum permissible number. The number is set in MD 28150 $MC_MM_NUM_VDIVAR_ELEMENTS.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program or machine data.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

14769 [Channel %1:] Block %2 Spindle %3 Implicit auxiliary function %4 Buffer full

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number %4 = Auxiliary function number
Definitions:	A maximum of 5 auxiliary functions of type "M" may be entered in an NC block. The upper limit is the total of programmed and implicitly generated M auxiliary functions. Implicit auxiliary functions M19 and M70 are generated, if in MD35035 $MA_SPIND_FUNCTION_MASK, bit 19 has been set for M19 and/or bit 20 for M70. M19 is generated with SPOS and SPOSA depending on the configuration. The same applies to M70 and transition into axis operation. The address extension corresponds to the spindle number like it is output to the PLC.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Distribute the M auxiliary functions and spindle functions that implicitly generate M19 and M70 to several blocks. - Deactivate any implicit auxiliary functions that are not required in MD35035 $MA_SPIND_FUNCTION_MASK, bit 19 and/or bit 20.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14770   [Channel %1:] Block %2 auxiliary function programmed incorrectly

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

Definitions: The permissible number of programmed auxiliary functions per NC block has been exceeded or more than one auxiliary function of the same auxiliary function group has been programmed (M and S function).
In the user-defined auxiliary functions, the maximum number of auxiliary functions per group in the NCK system settings has been defined for all auxiliary functions by means of the MD11100 $MN_AUXFU_MAXNUM_GROUP_ASSIGN (default: 1)
For each user-defined auxiliary function to be assigned to a group, the assignment is effected through 4 channel-specific machine data.
Return jump from asynchronous subprogram with M02/M17/M30, whereby the M code is not alone in the block. This is impermissible if the asynchronous subprogram interrupts a block with WAITE, WAITM or WAITMC. Remedy: Program M02/M17/M30 alone in the block or replace via RET.
22010 AUXFU_ASSIGN_TYPE: type of auxiliary function, e.g. M
22000 AUXFU_ASSIGN_GROUP: required group
22020 AUXFU_ASSIGN_EXTENSION: any required extension
22030 AUXFU_ASSIGN_VALUE: function value

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

Remedy: Correct the part program - max. 16 auxiliary functions, max. 5 M functions per NC block, max. 1 auxiliary function per group.

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

14780   [Channel %1:] Block %2 unreleased option used (identification %3)

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

Definitions: A non-released option has been used in the block.
Identifier Brief description
  1   LaserCtrl option
  2   ClearCtrl option
  3   FeedAdapt option
  4   AaTOff option
  5   Tang option
  6   LeadCtab option
  7   ELG option
  8   Trafo5 option
  9   Traoem option
 10   Transmit option
 11   Tracon option
 12   Tracyl option
 13   Traang option
 14   Oscill option
 15   SynSpi option
 16   Repos option
 17   Spline option
 18   Involute option
 19   Poly option
 20   Compress option
 23   Masl option
 24   ExtLang or ExtLanguage option not activated
 25   TechCycle option
 26   Liftfast option
 27   ProgAccel option
 33   AllAsupSynact option
 34   CmdAxSpind option
 35   Mea2 option
 36   ProgAnaOut option

37	OptAaTOff option
41	MachineMaintenance option
42	PathFeedSAInput option
45	ElecTransfer option
46	Cut3D option
47	CDA option
48	Reserved: generic coupling option
49	Measuring cycles option
50	ForceControl option
51	ESR option
52	Contour handwheel option

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program, retrofit option. In this context, please compare the available option data and/or (if available) the license image of your controller.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14781   **[Channel %1:] Block %2 motion synchronous action: %3 non-released option was used (identification %4)**

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Fine ID
Definitions:	A non-released option has been used in the block. Identifier: see alarm 14780
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program, retrofit option. In this context, please compare the available option data and/or (if available) the license image of your controller.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14782   **[Channel %1:] Block %2 non-active function used (identification %3)**

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

Definitions:	A non-active function is used in the block
	Brief description of the identification
	    1 Transformation
	    2 H number of the tool

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Modify part program. - Activate function.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14800   **[Channel %1:] Block %2 programmed path speed less or equal to zero**

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

Definitions:	Zero or a negative F or FZ value has been programmed in conjunction with the G functions G93, G94, G95 or G96. The path velocity may be programmed in the range of 0.001 to 999 999.999 [mm/min, mm/rev, mm/tooth, deg/min, deg/rev] for the metric input system and 0.000 1 to 39 999.999 [inch/min, inch/rev, inch/tooth] for the inch input system.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program the path velocity (geometric sum of the velocity components of the geometry axes involved) within the limits given above.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14810 [Channel %1:] Block %2 negative axis speed programmed for positioning axis %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis
Definitions:	A negative feed (FA value) has been programmed for the displayed axis presently operating as a positioning axis. The positioning velocity may be programmed in the range 0.001 to 999 999.999 [mm/min, deg/min] for the metric input system and 0.000 1 to 39 999.999 9 [inch/min, inch/rev] for the inch input system.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program the positioning velocity within the limits given above.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14811 [Channel %1:] Block %2 Incorrect value range for programmed dynamic value of axis/ spindle %3, error no. %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis, spindle %4 = Error no.
Definitions:	A value outside the permissible input range of a programmable dynamic value was used. The following causes of error are possible: 1: The value programmed for the axis velocity with VELOLIM or VELOLIMA lies outside the permissible range. The permissible range for VELOLIM is from 1 to 100 percent and for VELOLIMA from 1 to 200 percent. 2: The value programmed for the axis acceleration with ACC or ACCLIMA lies outside the permissible range from 1 to 200 percent. 3: The value programmed for the axis jerk with JERKLIM or JERKLIMA lies outside the permissible range from 1 to 200 percent.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Adjust the value range in accordance with the Programming Guide.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14812 [Channel %1:] Block %2 SOFTA not available for axis %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis
Definitions:	SOFT is to be set as type of motion control for an axis. This is not possible because a bent acceleration characteristic has been selected for this axis via machine data.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program or machine data.

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

14815	**[Channel %1:] Block %2 negative thread pitch change programmed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A negative thread pitch change has been programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct the value assignment. The programmed F value should be greater than zero. Zero is allowed but has no effect.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14820	**[Channel %1:] Block %2 negative value for maximum spindle speed programmed with constant cutting speed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	For the function "Constant cutting speed G96" a maximum spindle speed can be programmed with the keyword LIMS=.... The values are in the range 0.1 - 999 999.9 [rev/min].
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program the maximum spindle speed for the constant cutting speed within the limits given above. The keyword LIMS is modal and can either be placed in front of or within the block that selects the constant cutting speed.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14824	**[Channel %1:] Block %2 conflict with GWPS**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The functions of constant grinding wheel surface speed GWPS and constant cutting speed G96 S... have been activated at the same time for a spindle.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14840	**[Channel %1:] Block %2 incorrect value range for constant cutting speed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The programmed cutting speed is not within the input range Input range metric: 0.01 to 9 999.99 [m/min] Input range inch: 0.1 to 99 999.99 [inch/min]
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program cutting speed under address S within the permissible range of values.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14850   **[Channel %1:] Block %2 changing the reference axis for a constant cutting speed not allowed**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The attempt was made via the SCC[AX] instruction to change the reference axis for a constant cutting speed. This is not allowed if the indicated axis is no geometry axis.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform authorized personnel/service. When programming SCC[AX] indicate a geometry axis known in the channel.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14860   **[Channel %1:] Block %2 Selection of the tool cutting rate not allowed. Cause %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Cause of the error
Definitions:	Selection of the cutting rate SVC is not permissible in the current state Causes of the problem: the following function is active. 1: Constant cutting rate G96, G961 or G962 active 2: SPOS/SPOSA/M19 (spindle positioning mode) active 3: M70/axis mode active 4: SUG active
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Activate the speed control mode for the spindle prior to programming SVC, for example with M3, M4 or M5.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14861   **[Channel %1] Block %2 SVC programmed, but no tool offset active**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Cutting velocity SVC programmed in the block, but no tool offset active.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Select an appropriate tool prior to the SVC instruction.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14862   **[Channel %1] Block %2 SVC has been programmed, but the radius of the active tool correction is zero**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A cutting velocity SVC has been programmed in the block, but the radius of the active tool offset is zero. The radius of the active tool offset consists of the offset parameters $TC_DP6, $TC_DP12, $TC_SCPx6 and $TC_ECPx6.
Reaction:	Correction block is reorganized. Local alarm reaction.

Interface signals are set.
Alarm display.

Remedy:	Select an appropriate tool offset with a positive tool radius prior to the SVC instruction.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14863   [Channel %1] Block %2 The programmed SVC value is zero or negative

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The programmed value for the cutting velocity SVC is zero or negative.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Program an SVC value larger than zero.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14900   [Channel %1:] Block %2 center point and end point programmed simultaneously

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When programming a circle by means of the opening angle, the circle center point was programmed together with the circle end point. This is too much information for the circle. Only one of the two points is allowed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Select the programming variant guaranteeing that the dimensions are definitely taken over from the workpiece drawing (avoidance of calculation errors).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14910   [Channel %1:] Block %2 invalid angle of aperture for programmed circle

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When programming a circle by means of the opening angle, a negative opening angle or an opening angle greater than or equal to 360 degrees has been programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program opening angle within the allowed range of values between 0.0001 and 359.9999 [degrees].
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

14920   [Channel %1:] Block %2 intermediate point of circle incorrect

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	When programming a circle by means of an intermediate point (CIP) all 3 points (initial, end and intermediate points) are on a straight line and the intermediate point (programmed by means of interpolation parameters I, J, K) is not located between the initial and end points. If the circle is the component of a helix, the specified number of turns (keyword TURN=...) determines further block processing: - TURN>0: alarm display because the circle radius is infinitely great. - TURN=0 and CIP specified between initial and end points. A straight line is generated between the initial and end points (without alarm message).
Reaction:	Correction block is reorganized.

Interface signals are set.
Alarm display.

| **Remedy:** | Locate the position of the intermediate point with the parameters I, J and K in such a way that it actually is located between the initial and end points of the circle or do not make use of this type of circle programming and instead program the circle with radius or opening angle or center point parameters. |
| **Program Continuation:** | Clear alarm with NC START or RESET key and continue the program. |

15030 [Channel %1:] Block %2 different measurement system settings

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	The INCH or METRIC instruction describes the system of measurement in which the data blocks have been read from the control. In order to prevent the incorrect interpretation of data intended for a particular system of measurement, a data block is only accepted if the above instruction matches the active system of measurement.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Change the system of measurement or load a data block which matches the system of measurement.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15100 [Channel %1:] Block %2 REORG abort caused by log file overflow

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	In order to synchronize the preprocessing run and the main run with REORG, the control accesses modification data which are maintained in a logfile. The alarm indicates that no more capacity is available in the logfile for the specified block in the channel.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Please inform the authorized personnel/service department. No remedial measures are available for the further execution of the current part program, however:
	1. Reduce log file size requirement by:
	Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops STOPRE.
	2. Increase the size of the logfile by means of the channel-specific machine data:
	MD28000 $MC_MM_REORG_LOG_FILE_MEM and
	MD 28010 $MC_MM_NUM_REORG_LUD_MODULES
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15110 [Channel %1:] Block %2 REORG not possible

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	In order to synchronize the preprocessing run and the main run with REORG, the control accesses modification data which are maintained in a logfile. The alarm indicates that no more capacity is available in the logfile for the specified block in the channel.
	The alarm message means that the logfile has been deleted in order to obtain additional memory for program reorganization. Consequently, it is no longer possible to REORG the preprocessing memory up to the next coincidence point.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. No remedial measures are available for the further execution of the current part program, however:
	1. Reduce log file size requirement by:
	Reducing the distance between the preprocessing and the main run via appropriate preprocessing stops (STOPRE).
	2. Increase the size of the logfile by means of the channel-specific machine data:

	Modify MD28000 $MC_MM_REORG_LOG_FILE_MEM and Modify MD MD28010 $MC_MM_NUM_REORG_LUD_MODULES
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

15120   If a power failure occurs now, the last data changed will be lost; index/buffer size = %1

Parameters:	%1 = Index/buffer size
Definitions:	Notification alarm. The alarm has no negative impact on the current machining. One of the system-internal data buffers, in which the last changed, buffered data are stored, has overflowed because the current data change rate is too high. The alarm warns that a spontaneous power failure in this situation (mains fault, disconnect the system from the power supply) would cause a loss of the immediately previously changed buffered data (tool data, parts programs, R parameters, GUDs,...) If the system is operated in an environment in which a power failure cannot occur, then the output of this alarm can be prevented via machine data MD18232 $MN_MM_ACTFILESYS_LOG_FILE_MEM[index] = 0. For information, parameter %1 specifies the index of the machine data, and the buffer size set there.
Reaction:	Alarm display.
Remedy:	If the alarm is present only sporadically, it can simply be regarded as a notification. The regular control behavior is not affected. If one does not want to or cannot eliminate the cause, then the alarm can be suppressed by setting MD11415 $MN_SUPPRESS_ALARM_MASK_2; Bit3=1 ('H8'). If the alarm is permanently present, please inform the authorized personnel/service department. The value of MD18232 $MN_MM_ACTFILESYS_LOG_FILE_MEM[index] will then have to be increased.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

15122   Power ON after power failure: %1 data were restored, of which %2 machine data, %3 errors.

Parameters:	%1 = Number of data %2 = Number of machine data %3 = Number of errors occurred
Definitions:	Notification alarm. The alarm has no negative effect as long as %3 the number of errors occurred is zero. %1 indicates the number of elementary and complex data restoring steps which were taken after a power OFF during power ON orduring a power failure to restore the persistent NCK data. %2 indicates the number of restored machine data. If the value is larger than zero, another warm restart (NCK reset) may be necessary to make the - possibly configuring - machine data changes prior to the power failure effective. %3 indicates the number of errors occurred during data restoring.
Reaction:	Alarm display.
Remedy:	As long as %3 number of errors occurred is zero, the alarm is only informative. As long as %3 number of errors occurred is larger than zero, the alarm indicates a software error. Further machining with the data is not recommended. Please load a suitable archive file before continuing machining to avoid subsequent problems. Please inform the authorized personnel/service department. File /_N_MPF_DIR/_N_SIEMDIAGMEMPF_MPF contains information that may help Siemens for error diagnosis.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15150   [Channel %1:] Block %2 reload from external aborted

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

Definitions:	Execution from external was aborted because the reload buffer does not have enough machine function blocks (traversing blocks, auxiliary function, dwell time etc.). Background: When already executed machine function blocks are released, memory becomes available in the reload buffer. If machine function blocks are no longer released, nothing can be reloaded - this results in a deadlock situation. Example: Definition of extremely long curve tables via execution from external.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Insert machine function blocks in the part progam. - Increase the size of the reload buffer (MD18360 $MN_MM_EXT_PROG_BUFFER_SIZE). - Decrease the size of the curve table (Note: Blocks within CTABDEF/CTABEND are not machine function blocks).
Program Continuation:	Clear alarm with the RESET key. Restart part program

15160 [Channel %1:] Block %2 wrong preprocessing configuration

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A block element is required, but the block element memory is empty.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify the block search configuration in MD28060 $MC_MM_IPO_BUFFER_SIZE (decrease size of IPO buffer if necessary) or MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15165 [Channel %1:] Block %2 error when translating or interpreting Asup %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = String
Definitions:	At part program start and at start of an ASUB under Reset condition, the relevant data of all the ASUBs that can be activated at that time are processed: - PLC ASUBs - With MD20108 $MC_PROG_EVENT_MASK configured event-controlled program calls - ASUB after block search (MD11450 $MN_SEARCH_RUN_MODE bit 1=1) - Editable system ASUB ($MN_ASUP_EDITABLE) If an error occurs (converter or interpreter), alarm 15165 will be output first and then a converter or interpreter alarm that describes more details of the error. Alarm 15165 will cause an interpreter stop. A compensation block will not be possible.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15166 [Channel %1:] User system asup _N_ASUP_SPF not available

Parameters:	%1 = Channel number
Definitions:	By means of the MD11610 $MN_ASUP_EDITABLE the function "User-defined system asup" has been activated. However, the relevant user program could not be found in the specified search path: - 1. /_N_CUS_DIR/_N_ASUP_SPF - 2. /_N_CMA_DIR/_N_ASUP_SPF The default system asups are used.

Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Load the user-defined system asup in /_N_CUS_DIR/_N_ASUP_SPF or /_N_CMA_DIR/_N_ASUP_SPF laden.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15170   [Channel %1:] Block %2 program %3 could not be compiled

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = String
Definitions:	An error has occurred in compile mode. The (compiler) error message refers to the program specified here.
Reaction:	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the Delete key or NC START.

15175   [Channel %1:] Block %2 program %3. Interfaces could not be built

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = String
Definitions:	An error has occurred in interface generation mode. The (compiler) error message refers to the program specified here. In particular when loading new cycle program on the NCK, problems can occur if the value settings in MD18170 $MN_MM_NUM_MAX_FUNC_NAMES and MD18180 $MN_MM_NUM_MAX_FUNC_PARAM are too small.
Reaction:	Alarm display.
Remedy:	- Modify part program.
	- If new cycle programs have been loaded on the NCK, you will normally need to increase the values of MD18170 $MN_MM_NUM_MAX_FUNC_NAMES and MD18180 $MN_MM_NUM_MAX_FUNC_PARAM. See also the explanations for alarm 6010.
Program Continuation:	Clear alarm with the Delete key or NC START.

15176   [Channel %1:] Block %2 Program%3 may only be executed after Power ON

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = File name
Definitions:	If an encrypted program is loaded to the NCK, an NCK reset (restart) must be performed afterwards, because internal data for efficient processing of the encrypted program are preprocessed during NCK power-up. On calling an encrypted NC program it has now been detected that these data do not exist or are obsolete compared to the current version of the encrypted NC program.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Perform an NCK reset (restart).
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15177   [Channel %1:] Block %2 Error on preprocessing of program %3, error code: %4

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = File name
	%4 = Error code

Definitions:	If an encrypted program is loaded to the NCK, an NCK reset (restart) must be performed afterwards, because internal data for efficient processing of the encrypted program are preprocessed during NCK power-up. The following problem has occurred: Error code 1: Error on read-in of program %4 Error code 2: There is not enough DRAM memory available for storing the preprocessed data.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Error code 1: Encrypt and load program %4 again. Then perform an NCK reset (restart).
	Error code 2: Increase system SL 710-740, 802D, 828D: $MN_MM_T_FILE_MEM_SIZE.
	        Increase system SL 840 DI: $MN_MM_DRAM_FILE_MEM_SIZE.
	        Then perform an NCK reset (restart).

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

15180   [Channel %1:] Block %2: Error on editing program %3 as INI/DEF file.

Parameters:	%1 = Channel number %2 = Block number, label %3 = String
Definitions:	Errors were found when processing an initialization program (INI file), or a GUD or macro definition file (DEF file). The error message which is then displayed refers to the program specified here.
Reaction:	Alarm display.
Remedy:	Correct the initialization program (INI file), or the GUD or macro definition file (DEF file). In connection with Alarm 12380 or 12460, also change the memory configuration.
Program Continuation:	Clear alarm with the Delete key or NC START.

15182   [channel %1:] cycle alarm from the modified SIEMENS cycle %3

Parameters:	%1 = Channel number %2 = Is not used %3 = Path and file name of the modified SIEMENS cycle
Definitions:	When executing a SIEMENS cycle modified by the user, a cycle alarm was output with SETAL() (see follow-up alarm in the alarm output). Since the SIEMENS cycle was modified by the user (e.g. machine manufacturer), the cause for the cycle alarm must be determined / eliminated by the user who modified the cycle.
Reaction:	Interface signals are set. Alarm display.
Remedy:	The error cause leading to the cycle alarm cannot be investigated by SIEMENS since the know-how of the modified cycle sequence is with those who are responsible for the cycle change.
Program Continuation:	Clear alarm with the Delete key or NC START.

15185   [Channel %1:] %2 errors in INI file

Parameters:	%1 = Channel number %2 = Number of detected errors
Definitions:	Errors were found when processing initialization program _N_INITIAL_INI. This alarm will also be output, if errors are found during editing of _N_INITIAL_INI in the GUD definition files or if errors are found on ramp-up in the macro definition files.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Correct the INI or DEF file or correct the MD and create a new INI file (via "Upload").
Program Continuation:	Switch control OFF - ON.

15186 **[Channel %1:] %2 errors in GUD, macro or INI file**

Parameters:	%1 = Channel number
	%2 = Number of detected errors
Definitions:	%2 errors were found when processing GUD/macro definition files (DEF files) or initialization files (INI files)
	Alarm 15180 has already informed about the corresponding file.
	Prior to that the errors shown were reported by error-specific alarms, e.g. 12080 "syntax error".
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
Remedy:	Modify definition file or initialization file
Program Continuation:	Clear alarm with the RESET key. Restart part program

15187 **[Channel %1:] Error during execution of PROGEVENT file %3.**

Parameters:	%1 = Channel number
	%2 = Is not used
	%3 = PROGEVENT file name
Definitions:	An error has occurred on executing PROGEVENT.
	With alarm 15187, the name of the program that was started as PROGEVENT
	is displayed. Alarm 15187 is displayed together
	with the alarm that describes the error cause. Alarm 15187 is also output
	when the alarm occurs in a subroutine started from PROGEVENT.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Correct the PROGEVENT file (subroutine).
Program Continuation:	Clear alarm with the Delete key or NC START.

15188 **[Channel %1:] Error during execution of ASUB file %3.**

Parameters:	%1 = Channel number
	%2 = Is not used
	%3 = ASUB file name
Definitions:	An error has occurred on executing an ASUB.
	Alarm 15188 displays the name of the program that was started
	as ASUB. Alarm 15188 is output together with the alarm that
	describes the error cause. Alarm 15188 is also output when
	the alarm occurs in a subroutine started from the ASUB.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Correct the ASUB file (subroutine).
Program Continuation:	Clear alarm with the Delete key or NC START.

15189 **[Channel %1:] Error executing SAFE.SPF**

Parameters:	%1 = Channel number
Definitions:	An error occurred while processing the NC initialization program for Safety Integrated /_N_CST_DIR/_N_SAFE_SPF.
	This alarm is output together with the alarm describing the cause of the error.
Reaction:	Interpreter stop
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Correct /_N_CST_DIR/_N_SAFE_SPF and perform an NCK reset.

Program Continuation:	Switch control OFF - ON.

15190 [Channel %1:] Block %2 not enough free memory for subroutine call

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The following deadlock has been found in the interpreter: Memory is needed for calling a subroutine. The module memory is, however, empty and there is no prospect of module memory becoming free again by executing the preprocessing/main run queue, because this queue is empty.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Increase machine data MD28010 $MC_MM_NUM_REORG_LUD_MODULES / MD28040 $MC_MM_LUD_VALUES_MEM / MD18210 $MN_MM_USER_MEM_DYNAMIC or program a preprocessing stop STOPRE before calling the subroutine.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15300 [Channel %1:] Block %2 invalid number-of-passed-blocks during block search

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the function "Block search with calculation" a negative number of passes has been entered in column P (number of passes). The permissible range of values is P 1 - P 9 999.
Reaction:	Alarm display.
Remedy:	Enter only positive number of passes within the range of values.
Program Continuation:	Clear alarm with the Delete key or NC START.

15310 [Channel %1:] Block %2 file requested during block search is not available

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	During block search, a target has been specified with a program that has not been loaded.
Reaction:	Alarm display.
Remedy:	Correct the specified search target accordingly or reload the file.
Program Continuation:	Clear alarm with the Delete key or NC START.

15320 [Channel %1:] Block %2 invalid block search command

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The block search command (type of search target) is smaller than 1 or greater than 5. It is entered in column type of the block search window. The following block search orders are allowed. Type Meaning 1 Search for block number 2 Search for label 3 Search for string 4 Search for program name 5 Search for line number in a file
Reaction:	Alarm display.
Remedy:	Modify the block search command.
Program Continuation:	Clear alarm with the Delete key or NC START.

15330	**[Channel %1:] Block %2 invalid block number as search target**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Syntax error! Positive integers are allowed as block numbers. Block numbers must be preceded by ":" and subblocks by an "N".
Reaction:	Alarm display.
Remedy:	Repeat the input with corrected block number.
Program Continuation:	Clear alarm with the Delete key or NC START.

15340	**[Channel %1:] Block %2 invalid label as search target**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Syntax error! A label must have at least 2 but no more than 32 characters, and the first two characters must be alphabetic or underscore characters. Labels must be concluded with a colon.
Reaction:	Alarm display.
Remedy:	Repeat the input with corrected label.
Program Continuation:	Clear alarm with the Delete key or NC START.

15350	**[Channel %1:] Block %2 search target not found**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The specified program has been searched to the end of the program without the selected search target having been found.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Check the part program, change the block search (typing error in the part program) and restart the search.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15360	**[Channel %1:] Illegal target of block search (syntax error)**
Parameters:	%1 = Channel number
Definitions:	The specified search target (block number, label or string) is not allowed in block search.
Reaction:	Alarm display.
Remedy:	Correct target of block search.
Program Continuation:	Clear alarm with the Delete key or NC START.

15370	**[Channel %1:] Target of block search not found**
Parameters:	%1 = Channel number
Definitions:	In a block search, an impermissible search target has been specified (e.g. negative block number).
Reaction:	Alarm display.
Remedy:	Check the specified block number, label or character string. Repeat entry with correct search target.
Program Continuation:	Clear alarm with the Delete key or NC START.

15380 [Channel %1:] Block %2 illegal incremental programming in axis %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis
Definitions:	The first axis programming after "search to block end point" is performed incrementally. This is not allowed in the following situations: - After searching the target a transformation change has taken place. - A frame with rotation component is active. The programmed axis is involved in the rotation.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Find search destination in which the axes are programmed using an absolute reference. Deactivate adding of the accumulated search position with SD42444 $SC_TARGET_BLOCK_INCR_PROG = FALSE. Use search run with calculation "at contour".
Program Continuation:	Clear alarm with the RESET key. Restart part program

15400 [Channel %1:] Block %2 selected initial INIT block does not exist

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The operator has selected an INI block for a read, write or execution function which: 1. Does not exist in the NCK range or 2. Does not have the necessary protection level required for performing the function.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check whether the selected INI block is contained in the file system of the NCK. The present protection level must be selected to be at least equal to (or greater than) the protection level that has been defined for the read, write or execution function at the time of creating the file.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15410 [Channel %1:] Block %2 initialization file contains invalid M function

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The only M function allowed in an Init block is the M02, M17 or M30 end-of-program function.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Remove all M functions from the Init block except for the end identifier. An Init block may contain value assignments only (and global data definitions if they are not defined again in a program that can be executed later) but no motion or synchronous actions.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15420 [Channel %1:] Block %2 instruction in current mode not allowed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The alarm is output in the following situations: - The interpreter has detected an illegal instruction (e.g. a motion command) while processing an INI file or a definition file (macro or GUD). - In a GUD file, the access security for a machine data item is to be changed with REDEF, although an ACCESS file (_N_SACCESS_DEF, _N_MACCESS_DEF, _N_UACCESS_DEF) is available. Access rights for machine data can then only be changed via one of the ACCESS files with REDEF.

- When processing the safety initialization program /_N_CST_DIR/_N_SAFE_SPF an illegal instruction was detected due to the reduced language scope configured.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Correct the INI, GUD or macro file. - Correct part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15460 [Channel %1:] Block %2 syntax error when locking

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The addresses programmed in the block are not compatible with the modal syntax-determining G function. Example: N100 G01 ... I .. J.. K.. LF
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct the displayed block and ensure that the G functions and addresses in the block are in agreement.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15500 [Channel %1:] Block %2 illegal angle of shear

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The function CSHEAR has been called with an illegal (impossible) angle of shear, e.g. when the sum of angles between the axis vectors is greater than 360 degrees.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Program the angle of shear in accordance with the geometrical conditions of the machine and workpiece system.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

15701 [Channel %1:] Block %2 motion synchronous action: %3 illegal cycle alarm number %4

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Cycle alarm number
Definitions:	A SETAL command has been programmed with a cycle alarm number smaller than 60 000 or greater than 69 999. Alarm reaction of Siemens standard cycles: Nos. 61 000 -61 999: Interpreter stop; delete with Reset Nos. 62 000 - 62 999: Compensation block; delete with NC Start
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Program alarm number in the SETAL instruction in the correct range.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15800	**[Channel %1:] Block %2 wrong starting conditions for CONTPRON/CONTDCON**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	There is an error in the start conditions for CONTPRON/CONDCON: - G40 not active - SPLINE or POLY active - Unknown machining type programmed - Transferred machining direction not defined - Definition of LUDs in incorrect subroutine level - Transferred circle coordinates
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15810	**[Channel %1:] Block %2 wrong array dimension for CONTPRON/CONTDCON**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The number of columns for the array created for CONTPRON/CONTDCON does not conform to the current programming guide.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

15900	**[Channel %1:] Block %2 touch probe not allowed**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Measure with deletion of distance-to-go In the part program, an illegal probe has been programmed with the command MEAS (measure with deletion of distance-to-go). The probe numbers 0 ... no probe 1 ... probe 1 2 ... probe 2 are allowed, whether the probe is actually connected or not. Example: N10 MEAS=2 G01 X100 Y200 Z300 F1000 Probe 2 with deletion of distance-to-go
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Include a probe number within the limits given above in the keyword MEAS=... This must correspond to the hardware connection of the probe.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15910	**[Channel %1:] Block %2 touch probe not allowed**
Parameters:	%1 = Channel number %2 = Block number, label

Definitions:	Measure without deletion of distance-to-go
	In the part program, an illegal probe has been programmed with the command MEAW (measure without distance-to-go). The probe numbers
	0 ... no probe
	1 ... probe 1
	2 ... probe 2
	are allowed, whether the probe is actually connected or not.
	Example:
	N10 MEAW=2 G01 X100 Y200 Z300 F1000
	Probe 2 without deletion of distance-to-go
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Include a probe number within the limits given above in the keyword MEAW=... This must correspond to the hardware connection of the probe.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15950   [Channel %1:] Block %2 no traverse motion programmed

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Measure with deletion of distance-to-go
	In the part program, no axis or a traversing path of zero has been programmed with the command MEAS (measure with deletion of distance-to-go).
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct the part program and add the axis address or the traversing path to the measurements block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

15960   [Channel %1:] Block %2 no traverse motion programmed

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	Measure without deletion of distance-to-go
	In the part program, no axis or a traversing path of zero has been programmed with the command MEAW (measure without deletion of distance-to-go).
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct the part program and add the axis address or the traversing path to the measurements block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16000   [Channel %1:] Block %2 invalid value for lifting direction

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	During the "rapid lift from contour" (keyword: LIFTFAST), a code value for the lifting direction (keyword: ALF=...) which lies outside the permissible range (permissible value range: 0 to 8) was programmed .
	With active cutter radius compensation:
	Code numbers 2, 3 and 4 cannot be used in G41
	Code numbers 6, 7 and 8 cannot be used in G42 because they code the direction to the contour.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Program the lifting direction under ALF=... within the permissible limits.

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

16005   [Channel %1:] Block %2 invalid value for lifting distance

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Mistake in programming: the value for the lifting path must not be negative.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16010   [Channel %1:] Block %2 machining stop after lift fast

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	LIFTFAST without interrupt routine (Asup) has been programmed. The channel is stopped after the lift motion has been carried out.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	After the channel stop, the axes must be retracted manually in JOG and the program aborted with Reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16015   [Channel %1:] Block %2 wrong axis identifier %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	Axis names from different coordinate systems were used to program axes for LIFTFAST. The retraction movement is no longer clear.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Use axis names from one coordinate system.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16016   [Channel %1:] Block %2 no retraction position programmed for axis %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	The retraction enable was programmed for LIFTFAST without defining a retraction position for the axis. The retraction movement is no longer clear.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program a retraction position for the relevant axis.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16017	**[Channel %1:] Axis %2 Identifier %3, LIFTFAST ignores this axis, current axis not capable of retraction**
Parameters:	%1 = Channel %2 = Axis, spindle %3 = Identifier
Definitions:	LIFTFAST cannot be applied to the axis. Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 11. Identifiier (parameter 3) is bit-coded and displays some possible causes for the alarm: 0x01Axis is in another channel 0x02Axis is in spindle mode (e.g. SPOS) 0x04Axis is PLC axis 0x08Axis is oscillating axis 0x10Axis is neutral axis 0x20Axis is coupled slave axis 0x40Axis is in static synchronized action Overview of reactions of common programming to LIFTFAST: Axis | Synact | Reaction to LIFTFAST --- Path | | STOP + LIFTFAST POS | | STOP + LIFTFAST POS | non-modal | STOP + LIFTFAST POS | modal | STOP + LIFTFAST POS | stati. | RUN + SHOWALARM 16017 POSA | | STOP + LIFTFAST MOV | non-modal | STOP + LIFTFAST MOV | modal | STOP + LIFTFAST MOV | stati. | RUN + SHOWALARM 16017 PLC | | RUN + SHOWALARM 16017 Oscill. | | RUN + SHOWALARM 16017 SPOS | | STOP + SHOWALARM 16017 SPOS | non-modal | STOP + SHOWALARM 16017 SPOS | modal | STOP + SHOWALRAM 16017 SPOS | stati. | RUN + SHOWALARM 16017 SPOSA | | STOP + SHOWALARM 16017
Reaction:	Alarm display.
Remedy:	Remove axis from POLFMLIN or POLFMASK. Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 11. At the time of LIFTFAST, an axis for LIFTFAST is programmed, but the status of the axis does not allow LIFTFAST (e.g. oscillating axis or spindle), or the axis is not in the channel. LIFTFAST should only be applied to those axes that are capable of retracting at that time; POLFMASK or POLFMLIN should be adapted accordingly.
Program Continuation:	Clear alarm with the Delete key or NC START.

16020	**[Channel %1:] Repositioning in block %2 is not possible**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Programming or operator action incorrect: Repositioning via REPOS command is only possible in an asynchronous subprogram (interrupt routine). If the REPOS command was programmed, e.g. in the main program or in a cycle, part program execution is aborted with alarm 16020. In addition, the alarm is output in the following situations: - Access to $AC_RETPOINT (repositioning point) outside an ASUP (e.g. in the main program) - An axis to be repositioned was a oscillating axis with sychronous infeed (OSCILL) in the interrupted block and is now in a state that does not allow it to be traversed as a oscillating axis. Remedy: Change the axis to "neutral axis" state before repositioning with WAITP. - An axis to be repositioned was an infeed axis for a oscillating axis in the interrupted block; now it can no longer be traversed as one. Remedy: Change the axis back to "POS axis" state before repositioning.

Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Modify the part program if necessary.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16025 [Channel %1:] Block %2 impermissible axis exchange in REPOS command by axis %3.

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis identifier
Definitions:	With the REPOS command, an axis or spindle was programmed that was in the NEUTRAL state at that time. As the REPOS command cannot execute any implicit GET, these axes/spindles cannot be repositioned. Part program editing is therefore aborted.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Assign the axes/spindles that are to be repositioned to the channel via GET command prior to the REPOS command. Example: GET(A); assign the A axis to the channel REPOSL A; reposition the geometry axes and A axis
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16100 [Channel %1:] Block %2 spindle %3 not available in the channel

Parameters:	%1 = Channel number %2 = Block number, label %3 = String
Definitions:	Incorrect programming: This channel does not recognize the spindle number. The alarm can occur together with a dwell or a spindle function.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check the part program to determine whether the programmed spindle number is correct and whether the program is run in the correct channel. Check MD35000 $MA_SPIND_ASSIGN_TO_MACHAX for all machine axes to see whether one of them contains the programmed spindle number. This machine axis number must be entered in a channel axis of the machine data MD20070 $MC_AXCONF_MACHAX_USED.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16105 [Channel %1:] Block %2 spindle %3 cannot be assigned

Parameters:	%1 = Channel number %2 = Block number, label %3 = String
Definitions:	Mistake in programming: The programmed spindle is not assigned a real spindle by the spindle number converter. The alarm can be issued after improper use of SD42800 $SC_SPIND_ASSIGN_TAB[].
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Correct setting data or modify part program.

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

---

16111   [Channel %1:] Block %2 spindle %3 No speed programmed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle
Definitions:	Programming of a speed is expected.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program speed with S[spindle number]=..
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16200   [Channel %1:] Block %2 spline and polynominal interpolation not available

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The spline and polynomial interpolation are options that are not contained in the basic version of the control.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Do not program spline and polynomial interpolation, or retrofit the necessary option.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16410   [Channel %1:] Block %2 axis %3 is not a geometry axis

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	A geometry axis has been programmed that cannot be imaged on any machine axis in the current transformation (possibly there is no transformation active at the moment). Example: Without transformation: Polar coordinate system with X, Z, and C axis With transformation: Cartesian coordinate system with X, Y, and Z, e.g. with TRANSMIT.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Activate transformation type with TRAORI (n) or do not program geometry axes that do not participate in the transformation grouping.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16420   [Channel %1:] Block %2 axis %3 programmed repeatedly

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	It is not allowed to program an axis more than once.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Delete the axis addresses that have been programmed more than once.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16430 **[Channel %1:] Block %2 geometry axis %3 cannot traverse as positioning axis in rotated coordinate system**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	In the rotated coordinate system, traversing of a geometry axis as positioning axis (i.e. along its axis vector in the rotated coordinate system) would mean traversing of several machine axes. This is in conflict with the positioning axis concept, however, in which one axis interpolator runs in addition to the path interpolator!
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Traverse geometry axes as positioning axes only with rotation deactivated. Deactivate rotation: Keyword ROT without further specification of axis and angle. Example: N100 ROT
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16440 **[Channel %1:] Block %2 rotation programmed for non-existent geometry axis**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A rotation of a geometry axis which does not exist was programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16500 **[Channel %1:] Block %2 chamfer or rounding negative**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A negative chamfer or rounding has been programmed under the keywords CHF= ..., RND=... or RNDM=...
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Values for chamfers, roundings and modal roundings must be programmed with positive values only.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16510 **[Channel %1:] Block %2 no facing axis for diameter programming available**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Diameter programming has been activated although no transverse axis with diameter programming has been applied. Transverse axes can be applied with MD20100 $MC_DIAMETER_AX_DEF or MD30460 $MA_BASE_FUNCTION_MASK bit2 for diameter programming. Diameter programming can be applied through: - basic position DIAMON or DIAM90 of the G 29 group during booting - programming of DIAMON or DIAM90 - programming of DIAMONA[AX], DIAM90A[AX] or DAC, DIC, RAC, RIC
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department.
	When programming DIAMON/DIAM90, a traverse axis must be configured via MD20100 $MC_DIAMETER_AX_DEF.
	When programming DIAMONA[AX], DIAM90A[AX] or DAC, DIC, RAC, RIC, the AX axis must be a transverse axis for diameter programming configured via MD30460 $MA_BASE_FUNCTION_MASK bit2.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16600 [Channel %1:] Block %2 spindle %3 gear stage change not possible

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Spindle number
Definitions:	The programmed speed is outside the speed range of the set gear stage. In order to execute the programmed speed, the gear stage must be changed. In order to be able to execute the automatic gear stage change (M40 is active), the spindle must be in speed control operation.
	The alarm will no longer be output after having set bit 30 (0x40000000) in MD11410 $MN_SUPPRESS_ALARM_MASK. However, the function will not be affected by this.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	The changeover to speed control operation is performed by programming M3, M4 or M5. The M functions can be written together with the S word in the same block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16605 [Channel %1:] Block %2 Spindle %3 Gear stage change in %4 not possible

Parameters:	%1 = Channel number
	%2 = Block number, label
	%3 = Spindle number
	%4 = Gear stage
Definitions:	A gear stage change for the spindle will not be possible, if:
	- thread cutting (G33, G34, G35) is active
	- the spindle is active as master or slave spindle in a coupling
	- the spindle is being positioned
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	The gear stage is to be set prior to the corresponding machining step.
	If it is necessary, however, to change the gear stage within one of the above mentioned functions, this function must be switched off for the time of the gear stage change. Thread cutting is deselected with G1; synchronous spindle coupling is switched off with COUPOF; the spindle positioning operation is exited with M3, M4 or M5.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16687 [Channel %1:] Block %2 motion synchronous action %3 type of coupling/instruction %4 is not possible.

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
	%4 = Coupling type
Definitions:	The stated instruction is not permitted for the stated type of generic coupling.
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16700	**[Channel %1:] Block %2 axis %3 invalid feed type**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	In a thread cutting function, the feed has been programmed in a unit that is impermissible. G33 (thread with constant lead) and the feed have not been programmed with G94 or G95. G33 (thread with constant lead) is active (modal) and G63 is programmed additionally in a following block. (Conflict situation! G63 is in the 2nd G group, G33, G331 and G332 are in the 1st G group). G331 or G332 (rigid tapping) and the feed have not been programmed with G94.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Use only the feed type G94 or G95 in the thread cutting functions. After G33 and before G63, deselect the thread cutting function with G01.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16715	**[Channel %1:] Block %2 axis %3 spindle not in standstill**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number
Definitions:	In the applied function (G74, reference point approach), the spindle must be stationary.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program M5 or SPOS/SPOSA in front of the defective block in the part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16720	**[Channel %1:] Block %2 axis %3 thread pitch is zero**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	No pitch was programmed in a thread block with G33 (thread with constant pitch) or G331 (rigid tapping).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	The thread pitch must be programmed for the specified geometry axis under the associated interpolation parameters. X -> I Y -> J Z -> K
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16730	**[Channel %1:] Block %2 axis %3 wrong parameter**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	In G33 (tapping with constant pitch) the pitch parameter was not assigned to the axis that determines the velocity. For longitudinal and face threads, the thread pitch for the specified geometry axis must be programmed under the associated interpolation parameter. X -> I Y -> J Z -> K For taper threads, the address I, J, K depends on the axis with the longer path (thread length). A 2nd lead for the other axis is, however, not specified.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Assign lead parameters to the axis that determines the velocity.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16740   [Channel %1:] Block %2 no geometry axis programmed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	No geometry axis was programmed for tapping (G33) or for rigid tapping (G331, G332). The geometry axis is, however, essential if an interpolation parameter has been specified. Example: N100 G33 Z400 K2 ; thread pitch 2mm, thread end Z=400 mm N200 SPOS=0 ; position spindle in axis mode N201 G90 G331 Z-50 K-2 ; tapping to Z=-50, counterclockwise N202 G332 Z5 ; retraction, direction reversal automatic N203 S500 M03 ; spindle again in spindle mode
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Specify geometry axis and corresponding interpolation parameters.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16746   [Channel %1:] Block %2 spindle %3 selected gear stage %4 not installed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number %4 = Gear stage
Definitions:	The first gear stage data block is active. The required gear stage is not installed in the 1st gear stage data block. The number of gear stages installed is configured in MD35090 $MA_NUM_GEAR_STEPS. Examples of the occurrence of the alarm with 3 three gear stages installed (MD 35090 $MA_NUM_GEAR_STEPS = 3): * ... M44 or M45 has been programmed for the spindle concerned *...M70 has been programmed and MD35014 $MA_GEAR_STEP_USED_IN_AXISMODE is larger than 3.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program: Only those valid gear stages can be entered which have also been installed according to MD35090 $MA_NUM_GEAR_STEPS. Limit M70 configuration (MD 35014 $MA_GEAR_STEP_USED_IN_AXISMODE) to MD35090 $MA_NUM_GEAR_STEPS.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16747   [Channel %1:] Block %2 spindle %3 inserted gear stage %4 for tapping not installed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number %4 = Gear stage
Definitions:	The second gear stage data block has been activated for tapping with G331. However, the current gear stage has not been installed in the second gear stage data block. The number of gear stages installed is configured in MD35092 $MA_NUM_GEAR_STEPS2. The gear stage cannot be changed in traversing blocks. The gear stage appropriate for the speed must be loaded before the traversing block.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Procedure for automatically engaging the suitable gear stage prior to thread cutting: * Program the spindle speed (S) in a G331 block without axis motions and prior to thread cutting, e.g. G331 S1000. * Activate M40 for the spindle.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16748 [Channel %1:] Block %2 spindle %3 gear stage %4 expected

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number %4 = Gear stage
Definitions:	G331 activates the second gear stage data block for tapping. The programmed speed (S) of the master spindle lies outside the speed range of the active gear stage in the current traversing block. The gear stage cannot be changed in the traversing block. The gear stage appropriate for the speed must be loaded prior to the traversing block.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Procedure for automatically engaging the suitable gear stage prior to thread cutting: * Program the spindle speed (S) in a G331 block without axis motions and prior to thread cutting, e.g. G331 S1000. * Activate M40 for the spindle.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16750 [Channel %1:] Block %2 axis %3 SPCON not programmed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	For the programmed function (rotary axis, positioning axis), the spindle must be in position control mode.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program position control of the spindle with SPCON in the previous block.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16751 [Channel %1:] Block %2 spindle/axis %3 SPCOF not executable

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	For the programmed function, the spindle must be in the open-loop control mode. In the positioning or axis mode, the position control must not be deselected.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Put the spindle into open-loop control mode in the preceding block. This can be done with M3, M4 or M5 for the relevant spindle.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16755   **[Channel %1:] Block %2 no stop required**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	No Stop is needed for the programmed function. A Stop is necessary after SPOSA or after M5 if the next block is to be loaded only after a spindle stop.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Do not write instruction.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16760   **[Channel %1:] Block %2 axis %3 S value missing**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	No spindle speed has been given for rigid tapping (G331 or G332).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program the spindle speed under address S in [rpm] (in spite of axis mode); the direction of rotation is given by the sign of the spindle lead: - Positive thread pitch: Rotational direction as M03. - Negative thread pitch: Rotational direction as M04 N2.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16762   **[Channel %1:] Block %2 spindle %3 thread function is active**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number
Definitions:	Incorrect programming: the spindle function can currently not be executed. This alarm occurs when the spindle (master spindle) is linked with the axes by an interpolation function.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program. Deselect thread cutting or tapping.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16763   **[Channel %1:] Block %2 axis %3 programmed speed is illegal (zero or negative)**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	A spindle speed (S value) was programmed with the value zero or with a negative value.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	The programmed spindle speed (S value) must be positive. Depending on the application case, the value zero can be accepted (e.g. G25 S0).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16765	**[Channel %1:] Block %2 motion synchronous action: %3 following spindle/axis missing**
Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	The following spindle/axis has not been written in the part program.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16766	**[Channel %1:] Block %2 motion synchronous action: %3 string cannot be interpreted**
Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	A coupling has been switched on in which a non-interpretable string has been written (e.g. block change behavior).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16767	**[Channel %1:] Block %2 motion synchronous action: %3 leading spindle/axis missing**
Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	The master spindle/axis has not been programmed in the part program.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16770	**[Channel %1:] Block %2 axis %3 no measuring system available**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	One of the following spindle functions has been programmed, the position control requires: SPCON, SPOS, SPOSA, COUPON, G331/G332. The position control requires at least one measuring system. No measuring system has been configured in MD30200 $MA_NUM_ENCS of the programmed spindle.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. Retrofit a measuring system.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16775 [Channel %1:] Block %2 motion synchronous action: %3 axis %4 no measuring system available

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Axis name, spindle number
Definitions:	One of the following spindle functions has been programmed, the position control requires: SPCON, SPOS, SPOSA, COUPON, G331/G332. The position control requires at least one measuring system. No measuring system has been configured in MD30200 $MA_NUM_ENCS of the programmed spindle.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Retrofit a measuring system.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16800 [Channel %1:] Block %2 traverse instruction DC/CDC for axis %3 not allowed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	The keyword DC (Direct Coordinate) can only be used for rotary axes. This causes approach of the programmed absolute position along the shortest path. Example: N100 C=DC(315)
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Replace the keyword DC in the displayed NC block by specifying AC (Absolute Coordinate). If the alarm display is the result of an error in the axis definition, the axis can be declared as a rotary axis by means of the axis-specific MD30300 $MA_IS_ROT_AX. Corresponding machine data: Modify MD30310: $MA_ROT_IS_MODULO Modify MD30320: $MA_DISPLAY_IS_MODULO
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16810 [Channel %1:] Block %2 traverse instruction ACP for axis %3 not allowed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	The keyword ACP (Absolute Coordinate Positive) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. In the displayed NC block, replace the keyword ACP by specifying AC (Absolute Coordinate). If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD30300 $MA_IS_ROT_AX and MD30310 $MA_ROT_IS_MODULO can be declared a rotary axis with modulo change. Corresponding machine data: Modify MD30320 $MA_DISPLAY_IS_MODULO
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16820 [Channel %1:] Block %2 traverse instruction ACN for axis %3 not allowed

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	The keyword ACN (Absolute Coordinate Negative) is only allowed for "modulo axes". It causes approach of the programmed absolute position in the specified direction.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. In the displayed NC block, replace the keyword ACN by specifying AC (Absolute Coordinate). If the alarm display is based on an incorrect axis definition, the axis with the axis-specific machine data MD30300: $MA_IS_ROT_AX and MD30310: $MA_ROT_IS_MODULO can be declared a rotary axis with modulo change. Corresponding machine data: MD30320: $MA_DISPLAY_IS_MODULO
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16830 [Channel %1:] Block %2 incorrect position programmed for axis/spindle %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	A position beyond the range of 0 - 359.999 has been programmed for a modulo axis.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program position in the range 0 - 359.999.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16903 [Channel %1:] Program control: action %2<ALNX> not allowed in the current state

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The relevant action cannot be processed now. This can occur, for instance, during read-in of machine data.
Reaction:	Alarm display.
Remedy:	Wait until the procedure is terminated or abort with Reset and repeat the operation.
Program Continuation:	Clear alarm with the Delete key or NC START.

16904 [Channel %1:] Program control: action %2<ALNX> not allowed in the current state

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The operation (program, JOG, block search, reference point, etc.) cannot be started or continued in the current status.
Reaction:	Alarm display.
Remedy:	Check the program status and channel status.

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

16905   [Channel %1:] Program control: action %2<ALNX> not allowed

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Operation cannot be started or continued. A start is only accepted when an NCK function can be started. Example: A start is accepted in JOG mode when, for example, the function generator is active or a JOG movement has first been stopped with the Stop key.
Reaction:	Alarm reaction in Automatic mode.
Remedy:	Check the program status and channel status.
Program Continuation:	Clear alarm with the Delete key or NC START.

16906   [Channel %1:] Program control: action %2<ALNX> is aborted due to an alarm

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The action was aborted due to an alarm.
Reaction:	Alarm display.
Remedy:	Eliminate the error and acknowledge the alarm. Then repeat the operation.
Program Continuation:	Clear alarm with the Delete key or NC START.

16907   [Channel %1:] Action %2<ALNX> only possible in stop state

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action may only be performed in Stop state.
Reaction:	Alarm display.
Remedy:	Check the program status and channel status.
Program Continuation:	Clear alarm with the Delete key or NC START.

16908   [Channel %1:] Action %2<ALNX> only possible in reset state or at the block end

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action may only be performed in Reset state or at end of block. In JOG mode, no axis that is traversed as geometry axis in the switched coordinate system, must be active as PLC or command axis (started through static synchronized action) on mode change. This means that axes like that must be in the state 'neutral axis' again.
Reaction:	Alarm display.
Remedy:	Check the program status and channel status. Check in JOG mode whether the axes are PLC or command axes.
Program Continuation:	Clear alarm with the Delete key or NC START.

16909   [Channel %1:] Action %2<ALNX> not allowed in current mode

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	A different operating mode must be activated for the activated function.
Reaction:	Alarm display.
Remedy:	Check operation and operating state.
Program Continuation:	Clear alarm with the Delete key or NC START.

16911   **[Channel %1:] Mode change is not allowed**

Parameters:	%1 = Channel number
Definitions:	The change from overstoring into another operating mode is not allowed.
Reaction:	Alarm display.
Remedy:	After overstoring is terminated, it is possible to change to another operating state again.
Program Continuation:	Clear alarm with the Delete key or NC START.

16912   **[Channel %1:] Program control: action %2<ALNX> only possible in reset state**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action can only be performed in Reset state. Example: Program selection through HMI or channel communication (INIT) can only be performed in Reset state.
Reaction:	Alarm display.
Remedy:	Reset or wait until processing is terminated.
Program Continuation:	Clear alarm with the Delete key or NC START.

16913   **[Mode group %1:] [Channel %2:] Mode change: action %3<ALNX> not allowed**

Parameters:	%1 = Channel number %2 = Mode group number %3 = Action number/action name
Definitions:	The change to the desired mode is not permitted. The change can only take place in the Reset state. Example: Program processing is halted in AUTO mode by NC Stop. Then there is a mode change to JOG mode (program status interrupted). From this operating mode, it is only possible to change to AUTO mode and not to MDI mode!
Reaction:	Alarm display.
Remedy:	Either activate the Reset key to reset program processing, or activate the mode in which the program was being processed previously.
Program Continuation:	Clear alarm with the Delete key or NC START.

16914   **[Mode group %1:] [Channel %2:] Mode change: action %3<ALNX> not allowed**

Parameters:	%1 = Channel number %2 = Mode group number %3 = Action number/action name
Definitions:	Incorrect mode change, e.g.: Auto -> MDIREF.
Reaction:	Alarm display.
Remedy:	Check operation or selected mode.
Program Continuation:	Clear alarm with the Delete key or NC START.

16915   **[Channel %1:] Action %2<ALNX> not allowed in the current block**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	If traversing blocks are interrupted by asynchronous subroutines, then it must be possible for the interrupted program to continue (reorganization of block processing) after termination of the asynchronous subroutine. The 2nd parameter describes which action wanted to interrupt block processing.
Reaction:	Alarm display.
Remedy:	Let the program continue to a reorganized NC block or modify part program.
Program Continuation:	Clear alarm with the Delete key or NC START.

16916	**[Channel %1:] Repositioning: action %2<ALNX> not allowed in the current state**
Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Repositioning of block processing is presently not possible. A mode change cannot take place. The 2nd parameter describes which action should be used to perform repositioning.
Reaction:	Alarm display.
Remedy:	Let the program continue to a repositioned NC block or modify part program.
Program Continuation:	Clear alarm with the Delete key or NC START.

16919	**[Channel %1:] Action %2<ALNX> is not allowed due to a pending alarm**
Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action cannot be performed due to an alarm, or the channel is in the fault condition.
Reaction:	Alarm display.
Remedy:	Press the RESET key.
Program Continuation:	Clear alarm with the Delete key or NC START.

16920	**[Channel %1:] Action %2<ALNX> is already active**
Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	An identical action is still being processed.
Reaction:	Alarm display.
Remedy:	Wait until the previous procedure has been terminated and then repeat the operation.
Program Continuation:	Clear alarm with the Delete key or NC START.

16922	**[Channel %1:] Subprograms: action %2<ALNX> maximum nesting depth exceeded**
Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines are activated. These asynchronous subroutines can be interrupted in the same manner as user programs. Unlimited nesting depth is not possible for asynchronous subroutines due to memory limitations. Example: An interrupt interrupts the current program processing. Other interrupts with higher priorities interrupt processing of the previously activated asynchronous subroutines. Possible actions are: DryRunOn/Off, DecodeSingleBlockOn, delete distance-to-go, interrupts .....
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Do not trigger the event on this block.
Program Continuation:	Clear alarm with the RESET key. Restart part program

16923	**[Channel %1:] Program control: action %2<ALNX> not allowed in the current state**
Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The current processing cannot be stopped since a preprocessing process is active. This applies, for example, to the loading of machine data, and in block searches until the search target is found.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Cancel by pressing RESET!

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

16924 [Channel %1:] Caution: program test modifies tool management data

Parameters:	%1 = Channel number
Definitions:	Tool management data is changed during program testing. It is not possible to automatically rectify the data after termination of the program testing. This error message prompts the user to make a backup copy of the data or to reimport the data after the operation is terminated.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Save tool data on HMI and reimport data after "ProgtestOff".
Program Continuation:	Clear alarm with the Delete key or NC START.

16925 [Channel %1:] Program control: action %2<ALNX> not allowed in the current state, action %3<ALNX> active

Parameters:	%1 = Channel number %2 = Action number/action name %3 = Action number/action name
Definitions:	The action has been refused since a mode or sub-mode change (change to automatic mode, MDI, JOG, overstoring, digitizing, etc.) is taking place. Example: This alarm message is output if the Start key is pressed during a mode or sub-mode change from, for example, automatic to MDI, before the NCK has confirmed selection of the mode.
Reaction:	Alarm display.
Remedy:	Repeat action.
Program Continuation:	Clear alarm with the Delete key or NC START.

16927 [Channel %1:] Action %2<ALNX> at active interrupt treatment not allowed

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action may not be activated during interrupt processing (e.g. mode change).
Reaction:	Alarm display.
Remedy:	Reset or wait until interrupt processing is terminated.
Program Continuation:	Clear alarm with the Delete key or NC START.

16928 [Channel %1:] Interrupt treatment: action %2<ALNX> not possible

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	A program interrupt has been activated in a non REORG capable block. Examples of possible program interrupts in this case: - Traversing to fixed stop - Vdi channel delete distance-to-go - Vdi axial delete distance-to-go - Measuring - Software limit - Axis interchange - Axis comes from tracking - Servo disable - Gear stage change at actual gear stage unequal to setpoint gear stage. The block affected is a: - collection block from block search (except for the last collection block) - Block in overstore interrupt.

Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Do not trigger the event on this block.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16930 [Channel %1:] Preceding block and current block %2 must be separated through an executable block

Parameters:	%1 = Channel number
	%2 = Block number
Definitions:	The language functions WAITMC, SETM, CLEARM and MSG must be packed in separate NC blocks due to the language definition. To avoid velocity drops, these blocks are attached to the next NC block internally in the NCK (for MSG only in path control mode, for WAITMC to the previous NC block). For this reason, there must always be an executable block (not a calculation block) between the NC blocks. An executable NC block always includes e.g. travel movements, a help function, Stopre, dwell time etc.
Reaction:	Correction block is reorganized.
	Interpreter stop
	Interface signals are set.
	Alarm display.
Remedy:	Program an executable NC block between the previous and the current NC block.
	Example:
	N10 SETM.
	N15 STOPRE ; insert executable NC block.
	N20 CLEARM.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

16931 [Channel %1:] Subprograms: action %2<ALNX> maximum nesting depth exceeded

Parameters:	%1 = Channel number
	%2 = Action number/action name
Definitions:	Various actions can cause the current procedure to be interrupted. Depending on the action, asynchronous subroutines (ASUBs) are activated. These ASUBs can be interrupted in the same manner as the user program. Unlimited nesting depth is not possible for ASUBs due to memory limitations.
	Example: In the case of an approach block in a repositioning procedure do not interrupt repeatedly, instead wait until processing is completed.
	Possible actions are: mode change, SlashOn/Off, overstoring.
Reaction:	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Initiate a block change and repeat the action.
Program Continuation:	Clear alarm with the Delete key or NC START.

16932 [Channel %1:] Conflict when activating user data type %2

Parameters:	%1 = Channel number
	%2 = Data type
Definitions:	The "activate user data" function (PI service _N_SETUDT) modifies a data block (tool offset, settable work offset or base frame) which is also written by the NC blocks in preparation.
	In the event of a conflict, the value entered by the HMI is reset.
	Parameter %2 specifies which data block is affected:
	1: Active tool offset
	2: Base frame
	3: Active work offset
Reaction:	Alarm display.
Remedy:	Check the inputs on the HMI and repeat if necessary.

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

16933 [Channel %1:] Interrupt treatment: action %2<ALNX> not allowed in the current state

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	If a temporary standstill has occurred because of a Reorg event across block boundaries, it is possible that a block without Reorg capability has been loaded. In this situation, it is unfortunately necessary to abort the Reorg event handling! Reorg events are, e.g. abort subprogram, delete distance-to-go and interrupts.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Abort program with the RESET key.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16934 [Channel %1:] Interrupt treatment: action %2<ALNX> not possible due to stop

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Reorg events are, e.g. abort subprogram, delete distance to go and interrupts, axis exchange, termination of follow-up mode. Two Reorg events overlap in this situation. The 2nd Reorg event coincides with the 1st block generated by the previous event. (e.g. an axis exchange is induced twice in rapid succession). Axis exchange leads to Reorg in the channels in which an axis is removed without preparation. This block must be stopped in the above sequence in order to prevent the interpolator buffer from overflowing. This can be achieved by pressing the Stop or StopAll key, configuring an alarm with INTERPRETERSTOP or by decode single block.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	The program must be aborted with Reset.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

16935 [Channel %1:] Action %2<ALNX> not possible due to search run

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The action is not allowed as block search is currently running via program test. Block search via program test: "PI Service _N_FINDBL with mode parameter 5_. With this block search type, it is not permissible to activate program test or dry run feedrate.
Reaction:	Alarm display.
Remedy:	Activate the action after block search is terminated.
Program Continuation:	Clear alarm with the Delete key or NC START.

16936 [Channel %1:] Action %2<ALNX> not possible due to active dry run

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action is not allowed as dry run feedrate is currently active. Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when dry run feedrate is active.
Reaction:	Alarm display.
Remedy:	Abort program with the RESET key.
Program Continuation:	Clear alarm with the Delete key or NC START.

16937   **[Channel %1:] Action %2<ALNX> not possible due to program test**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	This action is not allowed as program test is currently active. Example: It is not permissible to activate block search via program test (PI service _N_FINDBL with mode parameter 5) when program test is active.
Reaction:	Alarm display.
Remedy:	Deactivate program test.
Program Continuation:	Clear alarm with the Delete key or NC START.

16938   **[Channel %1:] Action %2<ALNX> aborted due to active gear change**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Reorganization events are, among others, subprogram abort, delete distance-to- go and interrupts, axis exchange, exiting the correction state. These events wait for the end of a gear change. However, the maximum waiting period has elapsed.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Program must be aborted with Reset and, if necessary, MD10192 $MN_GEAR_CHANGE_WAIT_TIME must be increased.
Program Continuation:	Clear alarm with the RESET key. Restart part program

16939   **[Channel %1:] Action %2<ALNX> rejected due to active gear change**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Reorganization events that are possible in Stop state, e.g mode change, are waiting for the end of the gear change. However, the maximum waiting period has elapsed.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Repeat action or increase MD10192 $MN_GEAR_CHANGE_WAIT_TIME.
Program Continuation:	Clear alarm with the Delete key or NC START.

16940   **[Channel %1:] Action %2<ALNX> wait for gear change**

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Reorganization events are waiting for the end of a gear change. The alarm is displayed during the waiting period.
Reaction:	Alarm display. Warning display.
Remedy:	Alarm is suppressed by means of MD11411 $MN_ENABLE_ALARM_MASK bit 1 = 0.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

16941   **[Channel %1:] Action %2<ALNX> rejected because no program event has been executed yet**

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

Definitions:	The setting of the MD20108 $MC_PROG_EVENT_MASK forces an asynchronous subprogram to be triggered automatically on RESET or PowerOn. The implicitly triggered asynchronous subprograms are normally called "Event-triggered program call" or "Program event". In the alarm situation, this asynchronous subprogram could not yet be activated; that is why the action (normally start of part program) must be rejected. Reasons for the fact that the asynchronous subprogram could not be triggered: 1. The asynchronous subprogram does not exist (/_N_CMA_DIR/_N_PROG_EVENT_SPF) 2. The asynchronous subprogram is allowed to start in the referenced state only (see MD11602 $MN_ASUP_START_MASK) 3. READY is missing (because of alarm)
Reaction:	Alarm display.
Remedy:	Load program Check MD11602 $MN_ASUP_START_MASK Acknowledge alarm
Program Continuation:	Clear alarm with the Delete key or NC START.

16942   [Channel %1:] Start program command action %2<ALNX> not possible

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	Currently, the alarm occurs only in combination with the SERUPRO action. SERUPRO stands for search via program test. SERUPRO is currently searching the search target and has therefore switched this channel to the program test mode. With the START program command in channel 1, another channel 2 would actually be started, which means that axes would really be started during the search action. If this alarm is switched off (see help), the user can make use of the above behavior by initially selecting via PLC the program test mode in channel 2, leaving channel 2 executing until its natural end, stopping channel 2 in order to deselect program test again.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Alarm can be switched off with MD10708 $MN_SERUPRO_MASK bit 1.
Program Continuation:	Clear alarm with the RESET key. Restart part program

16943   [Channel %1:] Action %2<ALNX> not possible due to ASUP

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The action in the 2nd parameter was rejected, since an asynchronous subprogram is currently active. Currently, only the integrated search run is rejected with this alarm. The integrated search run is activated, if search run is triggered in the Stop program state. In other words: Parts of a program have already been executed and a following program part is "skipped" with search run in order to continue the program afterwards. The event is not possible if the program is stopped within an asynchronous subprogram or if an asynchronous subprogram had been selected before the event. An asynchronous subprogram is selected, when the triggering asynchronous subprogram event arrives, but the asynchronous subprogram cannot be started (e.g. the asynchronous start program is not started because of a read-in disable or because the Stop key is active). In this case, it is irrelevant whether a user ASUP or a system ASUP has been triggered. User ASUPs are activated via FC-9 or via the fast inputs. The following events lead to system ASUPS: - Mode change - Overstore on - Canceling subprogram level - Switching on of single block, type 2 - Setting machine data effective - Setting user data effective - Change skip levels - Dry run on/off - Program test off

	- Correction block alarms
	- Editing modi in Teach
	- External work offset
	- Axis exchange
	- Delete distance-to-go
	- Measuring
Reaction:	Alarm display.
Remedy:	Repeat the action after the end of the asynchronous subprogram.
Program Continuation:	Clear alarm with the Delete key or NC START.

16944 [Channel %1:] Action %2<ALNX> not possible due to active search blocks

Parameters:	%1 = Channel number
	%2 = Action number/action name
Definitions:	The NCK is currently processing either the action blocks of the search run or the approach motion after the search run. In this situation, the action (2nd parameter of the alarm) must be rejected. Currently, only the integrated search run is rejected with this alarm. The integrated search run is activated, if search run is triggered in the Stop program state. In other words: Parts of a program have already been executed and a following program part is "skipped" with search run in order to continue the program afterwards.
Reaction:	Alarm display.
Remedy:	Repeat the action after the approach motion of the search run.
Program Continuation:	Clear alarm with the Delete key or NC START.

16945 [Channel %1:] Action %2<ALNX> delayed up to the block end

Parameters:	%1 = Channel number
	%2 = Action number/action name
Definitions:	The currently executing action (e.g. dry run on/off, change skip levels, etc.) should be active immediately, but it can become active not earlier than at the end of the block, since a thread is currently being machined. The action is activated with a slight delay.
	Example: Dry run is started in the middle of the thread, then traversing at high speed does not start before the next block.
Reaction:	Alarm display.
Remedy:	Alarm can be switched off via MD11410 $MN_SUPPRESS_ALARM_MASK bit17==1.
Program Continuation:	Clear alarm with the Delete key or NC START.

16946 [Channel %1:] Start via START is not allowed

Parameters:	%1 = Channel ID
Definitions:	This alarm is active with "Group Serupro" only. "Group Serupro" is activated by means of MD10708 $MN_SERUPRO_MASK, Bit 2 and enables the retrace support of entire channel groups during block search.
	The MD22622 $MC_DISABLE_PLC_START specifies which channel is generally started from the PLC and which channel is only allowed to be started from another channel via the START part program command.
	This alarm occurs if the channel was started via the START part programm command and MD22622 $MC_DISABLE_PLC_START==FALSE was set.
Reaction:	Alarm display.
Remedy:	Modify MD22622 $MC_DISABLE_PLC_START of switch off "Group Serupro" (see MD10708 $MN_SERUPRO_MASK).
Program Continuation:	Clear alarm with the Delete key or NC START.

16947 [Channel %1:] Start via PLC is not allowed

Parameters:	%1 = Channel ID

Definitions:	This alarm is active with "Group Serupro" only. "Group Serupro" is activated by means of MD10708 $MN_SERUPRO_MASK, Bit 2 and enables the retrace support of entire channel groups during block search. The machine data MD22622 $MC_DISABLE_PLC_START specifies which channel is generally started from the PLC and which channel is only allowed to be started from another channel via the START part program command. This alarm occurs if the channel was started via the PLC and MD22622 $MC_DISABLE_PLC_START==TRUE was set.
Reaction:	Alarm display.
Remedy:	Modify MD22622 $MC_DISABLE_PLC_START of switch off "Group Serupro" (see MD10708 $MN_SERUPRO_MASK).
Program Continuation:	Clear alarm with the Delete key or NC START.

16948 [Channel %1:] dependent channel %2 still active

Parameters:	%1 = Channel ID %2 = Channel ID
Definitions:	This alarm is active with "Group Serupro" only. "Group Serupro" is activated by means of MD10708 $MN_SERUPRO_MODE, bit 2 and enables the retrace support of entire channel groups during block search. A _dependent channel_ is a channel that had indirectly been started by the currently active channel. The currently active channel was started via PLC. This channel m_u_s_t be terminated (i.e. reached M30) before the current channel is terminated. This alarm occurs if the currently active channel is terminated before the dependent channel.
Reaction:	Alarm display.
Remedy:	Switch off "Group Serupro" (see MD10708 $MN_SERUPRO_MASK) or install WAITE.
Program Continuation:	Clear alarm with the Delete key or NC START.

16950 [Channel %1:] Search run with hold block

Parameters:	%1 = Channel number
Definitions:	Informational alarm. The search run was not performed on the interruption block, instead, it touches down shortly before that. This so-called "hold block" is generated by the part program command IPTRLOCK, or implicitly defined by MD22680 $MC_AUTO_IPTR_LOCK. This is to prevent you from performing a search run in critical program areas (e.g. gear hobbing). The alarm also displays that, instead of searching for the block that actually was interrupted before, another block is being searched for. This behavior is desired and the alarm serves only informational purposes.
Reaction:	Alarm display.
Remedy:	MD11410 $MN_SUPPRESS_ALARM_MASK, MD22680 $MC_AUTO_IPTR_LOCK and language command IPTRLOCK
Program Continuation:	Clear alarm with the Delete key or NC START.

16951 [Channel %1:] Search run in a protected program section.

Parameters:	%1 = Channel number
Definitions:	A part programmer can define protected part program sections with the language commands IPTRLOCK and IPTRUNLOCK. Every search run in these program sections will then be acknowledged with alarm 16951. In other words: When the alarm appears, the user has started a search run (Serupro type) and the search target lies in a protected area. A protected area can also be defined implicitly with the MD22680 $MC_AUTO_IPTR_LOCK. Note: The alarm can only be generated if the simulation has been completed during the search run. The alarm cannot be output immediately at the start of the search run.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	MD11410 $MN_SUPPRESS_ALARM_MASK, MD22680 $MC_AUTO_IPTR_LOCK and language command IPTRLOCK
Program Continuation:	Clear alarm with the RESET key. Restart part program

16952	**[Channel %1:] Start program command not possible due to MDI**
Parameters:	%1 = Channel number
Definitions:	NCK is currently executing an ASUB in MDI mode. In this constellation, parts program command "Start" is not allowed for another channel. Attention: If an asup is started from JOG, the NCK can internally change to MDI, if the NCK was previously in MDI and not in RESET. Note: Without this alarm, the MDI buffer of the other channel would always be started.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Start ASUB in AUTO or ->JOG in AUTO
Program Continuation:	Clear alarm with the RESET key. Restart part program

16953	**[Channel %1:] For slave axis %2 SERUPRO not allowed, as master axis %3 not subject to axis/spindle disable**
Parameters:	%1 = Channel number %2 = Slave axis name, following spindle number %3 = Master axis name, master spindle number
Definitions:	Currently, the alarm occurs only in combination with the SERUPRO action. SERUPRO stands for search via program test. SERUPRO is possible only with an active coupling, if the axis/spindle disable is active for all master axes/spindles of the slave axis/spindle
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Set axis/spindle disable of the master axis
Program Continuation:	Clear alarm with the RESET key. Restart part program

16954	**[Channel %1:] Block %2 programmed stop prohibited in stop delay area**
Parameters:	%1 = Channel ID %2 = Block number, label
Definitions:	In a program area (stop delay area) that is bracketed with DELAYFSTON and DELAYFSTOF, a program command was used that causes a stop. No commands other than G4 are permissible that might cause a stop even though only shortly. A stop delay area can also be defined by MD11550 $MN_STOP_MODE_MASK.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	MD11550 $MN_STOP_MODE_MASK and language command DELAYFSTON DELAYFSTOF
Program Continuation:	Clear alarm with the RESET key. Restart part program

16955	**[Channel %1:] Stop in stop delay area is delayed**
Parameters:	%1 = Channel ID
Definitions:	In a program area (stop delay area) that is bracketed by DELAYFSTON and DELAYFSTOF, an event has been detected that causes a stop. The stop is delayed and executed after DELAYFSTOF. A stop delay area can also be defined by MD11550 $MN_STOP_MODE_MASK.
Reaction:	Interface signals are set. Alarm display.
Remedy:	MD11550 $MN_STOP_MODE_MASK and language command DELAYFSTON DELAYFSTOF

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

16956 [Channel %1:] Program %2 cannot be started due to global start disable.

Parameters:	%1 = Channel ID %2 = (path with program name)
Definitions:	The program selected in this channel cannot be started as "Global start disable" had been set. Note: PI "_N_STRTLK" sets the "Global start disable" and PI "_N_STRTUL" deletes the "Global start disable". The alarm is switched on with MD11411 $MN_ENABLE_ALARM_MASK bit 6.
Reaction:	Alarm display.
Remedy:	Delete the "Global start disable" and restart.
Program Continuation:	Clear alarm with the Delete key or NC START.

16957 [Channel %1:] Stop-Delay area is suppressed

Parameters:	%1 = Channel ID
Definitions:	The program area (Stop-Delay area), which is bracketed by DELAYFSTON and DELAYFSTOF, could not be activated. Every stop therefore becomes effective immediately and is not delayed! This occurs every time, when braking into a stop Stop-Delay area, i.e. a braking process starts before the Stop-Delay area and ends not earlier than in the Stop-Delay area. If the Stop-Delay area is entered with override 0, the Stop-Delay area can also not be activated Example: a G4 before the Stop-Delay area allows the user to reduce the override to 0. The next block in the Stop-Delay area then starts with override 0 and the alarm situation described occurs. MD11411 $MN_ENABLE_ALARM_MASK, bit 7 switches on this alarm.
Reaction:	Interface signals are set. Alarm display.
Remedy:	MD11550 $MN_STOP_MODE_MASK and language command DELAYFSTON DELAYFSTOF
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

16959 [Channel %1:] Action %2<ALNX> prohibited during simulation block search.

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The function (2nd parameter) must not be activated during simulation search.
Reaction:	Alarm display.
Remedy:	Wait for search end.
Program Continuation:	Clear alarm with the Delete key or NC START.

16960 [Channel %1:] Action %2<ALNX> prohibited during EXECUTE PROGRAM AREA.

Parameters:	%1 = Channel number %2 = Action number/action name
Definitions:	The function (2nd parameter) must not be activated during EXECUTE PROGRAM AREA.
Reaction:	Alarm display.
Remedy:	Wait for end of program area EXECUTE.
Program Continuation:	Clear alarm with the Delete key or NC START.

16961 [Channel %1:] Action %2<ALNX> prohibited during syntax check.

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

Definitions:	The function (2nd parameter) must not be activated during the syntax check. Comment: The syntax check is served by the following PI services: _N_CHKSEL _N_CHKRUN _N_CHKABO
Reaction:	Alarm display.
Remedy:	Wait for the end of the syntax check, or Cancel the syntax check with reset, or Cancel the syntax check with PI _N_CHKABO.
Program Continuation:	Clear alarm with the Delete key or NC START.

16962   [Channel %1:] NCK computing time reduced, start is not allowed.

Parameters:	%1 = Channel number
Definitions:	The computing time available to the NCK has been reduced, starts have therefore been locked. The computer performance is inadequate for smooth program execution. The computing time of the NCK may have been reduced by the HMI because of an HMI part program simulation.
Reaction:	Alarm display.
Remedy:	Wait for the simulation to end or press RESET in any channel.
Program Continuation:	Clear alarm with the Delete key or NC START.

16963   [Channel %1:] ASUB start declined

Parameters:	%1 = Channel number

Definitions:	An external ASUB start fom the ABORTED program state has been declined for the following reasons:
	  - Bit 0 is not set in MD11602 $MN_ASUP_START_MASK
	  - ASUB priority has been set too low or MD11604 $MN_ASUP_START_PRIO_LEVEL has been set too high

Reaction:	Alarm display.
Remedy:	Correct the machine data or change the priority of the ASUB to be executed.
Program Continuation:	Clear alarm with the Delete key or NC START.

16964   [Channel %1:] Executing of init blocks not fully completed

Parameters:	%1 = Channel number
Definitions:	Init blocks are processed during ramp-up; they ensure that the control is initialized correctly. The alarm is set if processing could not be completed correctly (usually due to alarms which were already pending).
Reaction:	Alarm display.
Remedy:	Eliminate pending alarms.
Program Continuation:	Switch control OFF - ON.

17000   [Channel %1:] Block %2 maximum number of symbols exceeded

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The maximum number of symbols defined by machine data MD28020 $MC_MM_NUM_LUD_NAMES_TOTAL has been exceeded.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. - Modify machine data - Reduce the number of symbols (variables, subroutines, parameters)
Program Continuation:	Clear alarm with the RESET key. Restart part program

17001 [Channel %1:] Block %2 no memory left for tool/magazine data

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

Definitions: The number of the following tool/magazine data variables in the NC is specified using machine data:
- Number of tools + number of grinding data blocks: MD18082 $MN_MM_NUM_TOOL
- Number of cutting edges: MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA
Tools, grinding data blocks, cutting edges can be used independently of the tool management.
The memory for the following data is available only if the corresponding bit has been set in MD18080 $MN_MM_TOOL_MANAGEMENT_MASK.
- Number of monitoring data blocks: MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA
- Number of magazines: MD18084 $MN_MM_NUM_MAGAZINE
- Number of magazine locations: MD18086 $MN_MM_NUM_MAGAZINE_LOCATION
The following variable is determined by the software configuration: Number of magazine spacing data blocks: P2 permits 32 such spacing data blocks.
Definition:
- 'Grinding data blocks': Grinding data can be defined for a tool of type 400 to 499. A data block of this type occupies as much additional memory as that provided for a cutting edge.
- 'Monitoring data blocks': Each cutting edge of a tool can be supplemented with monitoring data.
- If the alarm occurs while writing one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR, check whether machine data MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC / MD18076$MN_MM_NUM_LOCS_WITH_DISTANCE has been set correctly.
MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC defines the number of different Index1 statements that may be made for an Index2 value.
MD18076 $MN_MM_NUM_LOCS_WITH_DISTANCE defines the number of different buffer locations that may be named in Index2.
If a multitool is to be generated or its locations, the alarm indicates that either more multitools need to be generated than are permitted by the setting of MD18083 $MN_MM_NUM_MULTITOOL or, if the alarm occurs when the multitool locations are being generated, that more multitool locations need to be generated than is permitted by the setting of MD18085 $MN_MM_NUM_MULTITOOL_LOCATIONS.

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

Remedy: Please inform the authorized personnel/service department.
- Modify machine data
- Modify NC program, i.e. reduce number of rejected variable.

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

17010 [Channel %1:] Block %2 no memory left

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

Definitions: When executing/reading files from the active working memory, it was found that there is not enough memory space (e.g. for large multidimensional arrays or when creating tool offset memory).

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

Remedy: Please inform the authorized personnel/service department. Make arrays smaller or make more memory space available for memory management of subroutine calls, tool offsets and user variables (machine data MM_...).
See /FB/, S7 Memory Configuration

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

17020 [Channel %1:] Block %2 illegal array index 1

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

Definitions:	General:
	Read or write access has been programmed to an array variable with an illegal 1st array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
	If an MT number is programmed, the value may collide with a previously defined T number or a previously defined magazine number.
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct the specification of array elements in the access instruction to match the defined size. If an SPL is used in Safety Integrated, the field index via optional data may be subject to additional restrictions.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17030   [Channel %1:] Block %2 illegal array index 2

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	General:
	A read or write access has been programmed to an array variable with an invalid 2nd array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct the specification of array elements in the access instruction to match the defined size.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17035   [Channel %1:] Block %2 illegal array index 1

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	General:
	A read or write access has been programmed to an array variable with an invalid 3rd array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Correct the specification of array elements in the access instruction to match the defined size.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17040   [Channel %1:] Block %2 illegal axis index

Parameters:	%1 = Channel number
	%2 = Block number, label
Definitions:	A read or write access has been programmed to an axial variable in which the axis name cannot be unambiguously imaged on a machine axis.
	Example:
	Writing of an axial machine data
	$MA_... [X]= ... ; but geometry axis X cannot be imaged on a machine axis because of a transformation!
Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
Remedy:	Deselect transformation before writing into the axial data (keyword: TRAFOOF) or use the machine axis names as axis index.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17050   [Channel %1:] Block %2 illegal value

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

Definitions:	A value has been programmed that exceeds the value range or a limit value of a variable or a machine data item. For example: In a string variable (e.g. GUD or LUD), a string needs to be written that exceeds the agreed string length in the variable definition.
	  - If an illegal value is to be written to a tool or magazine management variable (e.g. illegal cutting edge number in $TC_DPCE[x,y] or illegal magazine location number in $TC_MDP2[x,y])
	   - An illegal value is to be written in $P_USEKT or $A_DPB_OUT[x,y]
	   - An illegal value is to be written in a machine data (e.g. MD10010 $MN_ASSIGN_CHAN_TO_MODE_GROUP[0] = 0)
	    - On accessing an individual frame element, a frame component other than TRANS, ROT, SCALE or MIRROR was addressed or the CSCALE function was assigned a negative scale factor.
	A multitool number has been programmed that collides with a previously defined T number or a previously defined magazine number.
	When programming DELMLOWNER: The command cannot be programmed with the T number of a tool that is part of a multitool.

Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Address frame components only with the keywords provided; program the scale factor between the limits of 0.000 01 to 999.999 99.
Program Continuation:	Clear alarm with the RESET key. Restart part program

17055   [Channel %1:] Block %2 GUD variable not existing

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The required GUD variable was not found for a MEACALC procedure during read or write access.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Check whether all the GUDs were created for MEACALC. DEF CHAN INT _MVAR, _OVI[11] DEF CHAN REAL _OVR[32], _EV[20], _MV[20], _SPEED[4], _SM_R[10], _ISP[3] DEF NCK REAL _TP[3,10], _WP[3,11], _KB[3,7], _CM[8], _MFS[6] DEF NCK BOOL _CBIT[16] DEF NCK INT _CVAL[4].
Program Continuation:	Clear alarm with the RESET key. Restart part program

17060   [Channel %1:] Block %2 requested data area too large

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The maximum memory space of 8 KB available for a symbol has been exceeded.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Reduce array dimensions.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17070	**[Channel %1:] Block %2 data is write-protected**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt was made to write into a write-protected variable (e.g. a system variable). Safety Integrated: Safety system variables can only be written into via the safety SPL program.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify part program. Safety Integrated: - Delete write accesses to safety system variables in part programs other than the safety SPL program - Verify the release of the safety functionality
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17071	**[Channel %1:] Block %2 data read-protected**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt has been made to read a read-protected variable (e.g. a system variable).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please set the corresponding access right or modify the part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17080	**[Channel %1:] Block %2 %3 value below lower limit**
Parameters:	%1 = Channel number %2 = Block number, label %3 = MD
Definitions:	An attempt was made to write into a machine data with a value smaller than the defined lower limit.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Determine the input limits of the machine data and assign a value within these limits.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17090	**[Channel %1:] Block %2 %3 value exceeds upper limit**
Parameters:	%1 = Channel number %2 = Block number, label %3 = MD
Definitions:	An attempt was made to write into a machine data with a value greater than the defined upper limit.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Determine the input limits of the machine data and assign a value within these limits.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17095 [Channel %1:] Block %2 invalid value

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt was made to write an invalid value, e.g. zero, into a machine data.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Correct the value assignment, e.g. a value within the value range not equal to zero.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17100 [Channel %1:] Block %2 digital input/comparator no. %3 not activated

Parameters:	%1 = Channel number %2 = Block number, label %3 = Input number
Definitions:	Either an attempt was made to read a digital input n via the system variable $A_IN[n] and this input has not been activated via NCK MD10350 $MN_FASTIO_DIG_NUM_INPUTS; or to read a comparator input via system variable $A_INCO[n] and this input belongs to a comparator which has not been activated.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify part program or machine data accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17110 [Channel %1:] Block %2 digital output no. %3 not activated

Parameters:	%1 = Channel number %2 = Block number, label %3 = No. of output
Definitions:	An attempt was made to read or set a digital NCK output (connector X 121) via the system variable $A_OUT [n] with the index [n] greater than the specified upper limit in MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program index [n] of the system variable $A_OUT [n] only between 0 and the value in MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17120 [Channel %1:] Block %2 analog input no. %3 not activated

Parameters:	%1 = Channel number %2 = Block number, label %3 = Input number
Definitions:	An attempt has been made by means of the system variable $A_INA[n] to read an analog input n that has not been activated by the MD10300 $MN_FASTIO_ANA_NUM_INPUTS.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify part program or machine data accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17130   **[Channel %1:] Block %2 analog output no. %3 not activated**

Parameters:	%1 = Channel number %2 = Block number, label %3 = No. of output
Definitions:	An attempt has been made by means of the system variable $A_OUTA[n] to write or read an analog output n that has not been activated by the MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify part program or machine data accordingly.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17140   **[Channel %1:] Block %2 NCK output %3 is assigned to a function via machine data**

Parameters:	%1 = Channel number %2 = Block number, label %3 = No. of output
Definitions:	The programmed digital/analog output is assigned to an NC function (e.g. software cams).
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Use another output or deactivate concurrent NC function via MD.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17150   **[Channel %1:] Block %2 maximum of %3 NCK outputs programmable in the block**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Quantity
Definitions:	No more than the specified number of outputs may be programmed in an NC block. The quantity of hardware outputs is defined in the MDs: MD10360 $MN_FASTIO_DIG_NUM_OUTPUTS and MD10310 $MN_FASTIO_ANA_NUM_OUTPUTS
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program fewer digital/analog outputs in a block. The specified maximum number applies in each case separately for analog or digital outputs. If necessary, program two NC blocks.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17160   **[Channel %1:] Block %2 no tool selected**

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt has been made to access the current tool offset data via the system variables: $P_AD [n]: Contents of the parameter (n: 1 - 25) $P_TOOL: Active D number (tool edge number) $P_TOOLL [n]: Active tool length (n: 1- 3) $P_TOOLR: Active tool radius although no tool had been selected previously.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Program or activate a tool offset in the NC program before using the system variables. Example: N100 G.. ... T5 D1 ... LF With the channel-specific machine data: Modify MD22550 $MC_TOOL_CHANGE_MODE New tool offset for M function Modify MD22560 $MC_TOOL_CHANGE_M_CODE M function with tool change It is established whether a tool offset is activated in the block with the T word or whether the new offset values are allowed for only when the M word for tool change occurs.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

17170   [Channel %1:] Block %2 number of symbols too large

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The predefined symbols could not be read in during power-up.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	--
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

17180   [Channel %1:] Block %2 illegal D number

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	In the displayed block, access is made to a D number that is not defined and therefore is not available.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check tool call in the NC parts program: - Correct tool correction number D programmed? If no D number is specified with the tool change command, then the D number set by MD20270 $MC_CUTTING_EDGE_DEFAULT will be active automatically. It is D1 by default. - Tool parameters (tool type, length,...) defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through a tool data file in NCK. Description of the system variables $TC_DPx[t, d] as included in a tool data file. x ... Correction parameter number P t ... Associated tool number T d ... Tool correction number D
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17181   [Channel %1:] Block %2 T no.= %3, D no.= %4 not existing

Parameters:	%1 = Channel number %2 = Block number, label %3 = T number %4 = D number
Definitions:	A programmed D number was not recognized by the NC. By default, the D number refers to the specified T number. If the flat D number function is active, T= 1 is output.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	If the program is incorrect, remedy the error with a correction block and continue the program. If the data block is missing, download a data block for the specified T/D values onto the NCK (via HMI with overstore) and continue the program.

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

17183   **[Channel %1:] Block %2 H number already available in T no.= %3, D no.= %4**

Parameters:	%1 = Channel number %2 = Block number, label %3 = T number %4 = D number
Definitions:	Each H number (except for H=0) must be assigned in a TO unit only once. The indicated edge already has the H number. If the H number shall be assigned more than once, MD10890 $MN_EXTERN_TOOLPROG_MODE, bit 3 must be set = 1.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Change program: - Select different H number
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17188   **[Channel %1:] D number %2 defined in tool T no. %3 and %4**

Parameters:	%1 = Channel number %2 = Offset number D %3 = T number of first tool %4 = T number of second tool
Definitions:	The specified D number %2 in the TO unit of channel %1 is not unique. The specified T numbers %3 and %4 each have an offset with number %2. If tool management is active: The specified T numbers belong to tool groups with different names.
Reaction:	Interface signals are set. Alarm display.
Remedy:	1. Ensure that the D numbers within the TO unit are unique. 2. If unique numbering is not necessary for subsequent operations, do not use the command.
Program Continuation:	Clear alarm with the Delete key or NC START.

17190   **[Channel %1:] Block %2 illegal T number %3**

Parameters:	%1 = Channel number %2 = Block number, label %3 = T number
Definitions:	In the displayed block, access is made to a tool that is not defined and therefore not available. The tool has been named by its T number, its name or its name and duplo number.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check tool call in the NC part program: - Correct tool number T.. programmed? - Tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through the V.24 interface. Description of the system variables $P_DP x [n, m] n ... Associated tool number T m ... Tool edge number D x ... Parameter number P
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17191	**[Channel %1:] Block %2 T= %3 not existing, program %4**
Parameters:	%1 = Channel number %2 = Block number, label %3 = T number or T identifier %4 = Program name
Definitions:	A tool identifier which the NCK does not recognize was programmed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	If the program pointer is at an NC block which contains the specified T identifier: If the program is incorrect, remedy the error with a correction block and continue the program. If the data block is missing, create one. You can do this by downloading a data block with all the defined D numbers onto the NCK (via MMC with overstore) and continue the program. If the program pointer is at an NC block which does not contain the specified T identifier: The error occurred at an earlier point in the program where the T command appeared, but the alarm was not output until the change command was detected. If the program is incorrect - T5 programmed instead of T55 - the current block can be corrected with a correction block; i.e. if only M06 is entered, you can correct the block with T55 M06. The incorrect T5 line remains in the program until it is terminated by a RESET or end of program. In complex program structures with indirect programming, it may not be possible to correct the program. In this case, you can only intervene locally with an overstore block - with T55 in the example. If the data block is missing, create one. You can do this by downloading the data block of the tool with all the defined D numbers onto the NCK (via MMC with overstore), program the T command with overstore, and continue the program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17194	**[Channel %1:] Block %2 no suitable tool found**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	- An attempt was made to access a tool which has not been defined. - The specified tool does not permit access. - A tool with the desired properties is not available.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check access to tool: - Are the parameters of the command correctly programmed? - Does the status of the tool prevent access?
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17195	**[Channel %1:] block %2 illegal tool holder number %3**
Parameters:	%1 = Channel number %2 = Block number, label %3 = Tool holder number
Definitions:	In the displayed block, a tool holder that is not defined is accessed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check the programming of the tool holder in the NC program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17200   [Channel %1:] Block %2: Data of tool %3 cannot be deleted.

Parameters:	%1 = Channel number %2 = Block number, label %3 = T number
Definitions:	An attempt has been made to delete from the part program the tool data for a tool currently being processed. Tool data for tools involved in the current machining operation may not be deleted. This applies both for the tool preselected with T or that has been changed in place of another, and also for tools for which the constant grinding wheel peripheral speed or tool monitoring is active.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Check access to tool offset memory by means of $TC_DP1[t,d] = 0 or deselect tool.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17210   [Channel %1:] Block %2 access to variable not possible

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The variable cannot be written/read directly from the part program. It is allowed only in motion synchronous actions. Example for variable: $P_ACTID (which planes are active) $AA_DTEPB (axial distance-to-go for reciprocating infeed) $A_IN (test input) Safety Integrated: Safety PLC system variables can only be read during the safety SPL setup phase.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17224   [Channel %1:] Block %2 tool T/D= %3 - tool type %4 is not permitted

Parameters:	%1 = Channel number %2 = Block number, label %3 = Incorrect T no. / D no. %4 = Incorrect tool type
Definitions:	On this system, it is not possible to select tool offsets of the indicated tool types. The variety of tool types can both be limited by the machine OEM and be reduced on individual control models. Only use tools of the tool types permitted for this system. Check whether an error has occurred on defining the tool.
Reaction:	Correction block is reorganized. Interpreter stop Interface signals are set. Alarm display.
Remedy:	Correct the NC program or correct the tool data
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17270   [Channel %1:] Block %2 call-by-reference: illegal variable

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Machine data and system variables must not be transferred as call-by-reference parameters.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Modify NC program: Assign the value of the machine data or of the system variable to a program-local variable and transfer this as parameter.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17500 [Channel %1:] Block %2 axis %3 is not an indexing axis

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	An indexing axis position has been programmed for an axis with the keywords CIC, CAC or CDC that has not been defined as indexing axis in the machine data.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Remove programming instruction for indexing axis positions (CIC, CAC, CDC) from the NC part program or declare the relevant axis to be an indexing axis. Indexing axis declaration: Modify MD30500: $MA_INDEX_AX_ASSIGN_POS_TAB (indexing axis assignment) The axis will become an indexing axis when an assignment to an indexing position table was made in the stated MD. 2 tables are possible (input value 1 or 2). Modify MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1 Modify MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2 (Number of positions for 1st/2nd indexing axis) Standard value: 0 Maximum value: 60 Modify MD10910 $MN_INDEX_AX_POS_TAB_1 [n] Modify MD10930 $MN_INDEX_AX_POS_TAB_2 [n] (Positions of the 1st indexing axis) The absolute axis positions are entered. (The list length is defined via MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1).
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17501 [Channel %1:] Block %2 indexing axis %3 with Hirth tool system is active

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	The 'Hirth tooth system' function is activated for the indexing axis. This axis can therefore approach only indexing positions, another travel movement of the axis is not possible.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct part program. Correct FC16 or FC18 call. Deselect machine data MD30505 $MA_HIRTH_IS_ACTIVE.
Program Continuation:	Clear alarm with the RESET key. Restart part program

17502 [Channel %1:] Block %2 indexing axis %3 with Hirth tooth system stop is delayed

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

Definitions:	For the indexing axis, the 'Hirth tooth system' function is activated and the override has been set to 0 or another stop condition (e.g. VDI interface signal) is active. Since it is possible to stop only on indexing axes, the next possible indexing position is approached. The alarm is displayed until this position is reached or the stop condition is deactivated.
Reaction:	Alarm display.
Remedy:	Wait until the next possible indexing position is reached or set override > 0 or deactivate another stop condition.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

17503 [Channel %1:] Block %2 indexing axis %3 with Hirth tooth system and axis not referenced

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	The 'Hirth tooth system' function is activated for the indexing axis and the axis is to be traversed although it is not referenced.
Reaction:	Alarm display.
Remedy:	Reference axis.
Program Continuation:	Clear alarm with the Delete key or NC START.

17505 [Channel %1:] Block %2 motion synchronous action: %3 indexing axis %4 is active with Hirth tooth system

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Axis name
Definitions:	The 'Hirth tooth system' function is activated for the indexing axis. This axis can therefore approach only indexing positions, another travel movement of the axis is not possible.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Correct part program. Correct FC16 or FC18 call. Deselect machine data MD30505 $MA_HIRTH_IS_ACTIVE.
Program Continuation:	Clear alarm with the RESET key. Restart part program

17510 [Channel %1:] Block %2 invalid index for indexing axis %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	The programmed index for the indexing axis is beyond the position table range. Example: Perform an absolute approach of the 56th position in the list allocated via the axis-specific MD30500 $MA_INDEX_AX_ASSIGN_POS_TAB with the 1st positioning axis, the number of positions is e.g. only 40 (MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1 = 40). N100 G.. U=CAC (56) Or, with equidistant distances, the programmed index is smaller or equal 0. Or, an attempt is made with a MOV movement to travel to a position outside the permitted area.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Program the indexing axis position in the NC part program in accordance with the length of the current position table, or add the required value to the position table and adjust the length of the list.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

17601 [Channel %1:] Block %2 motion synchronous action: %3 preset not possible on axis %4

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Axis name, spindle number
Definitions:	The actual value cannot be preset for this axis because the axis is in motion or the axis is linked in a transformation.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Avoid setting actual value.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17640 [Channel %1:] Block %2 spindle operation for transformed axis %3 not possible

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	The axis programmed for the spindle operation is involved in the current transformation as geometry axis. This is not allowed.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	First switch off the transformation function.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17800 [Channel %1:] Block %2 illegally coded position programmed

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The position number n specified with the keyword FP=n is not permissible. Two (2) absolute axis positions can be directly defined as fixed points by the axis-specific machine data MD30600 $MA_FIX_POINT_POS[n]. Or, if position numbers 3 and/or 4 are to be used, then machine data MD30610 $MA_NUM_FIX_POINT_POS must be set accordingly.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Program keyword FP with machine fixed points 1 or 2. Example: Approach fixed point 2 with machine axes X1 and Z2. N100 G75 FP=2 X1=0 Z2=0 Or modify MD30610 $MA_NUM_FIX_POINT_POS and, if necessary, MD30600 $MA_FIX_POINT_POS[].
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

17810 [Channel %1:] Axis %2 not referenced

Parameters:	%1 = Channel number %2 = Axis number
Definitions:	A function has been activated for the axis in JOG mode, e.g. fixed-point approach, JOG to position, JOG in circles, but the axis has not been referenced.

Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Reference axis.
Program Continuation:	Clear alarm with the Delete key or NC START.

17811   **[Channel %1:] Fixed-point approach not possible for axis %2 in JOG, reason %3**

Parameters:	%1 = Channel number
	%2 = Axis name, spindle number
	%3 = Cause
Definitions:	A 'fixed-point approach in JOG' has been requested for an axis. This is not possible because:
	Reason 1: The axis is involved in the active transformation.
	Reason 2: The axis is a following axis in an active coupling.
	The fixed point approach will therefore not be executed.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Deselect fixed-point approach in JOG, or previously deselect the transformation with TRAFOOF or disband the coupling.
Program Continuation:	Clear alarm with the Delete key or NC START.

17812   **[Channel %1:] Axis %2 fixed-point approach in JOG: Fixed point %3 changed**

Parameters:	%1 = Channel number
	%2 = Axis name, spindle number
	%3 = Fixed-point number
Definitions:	'Fixed-point approach in JOG' is active for the axis, but another fixed point has been selected, or the fixed-point approach has been deactivated. The approach motion is canceled.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17813   **[Channel %1:] Axis %2 fixed-point approach in JOG and override motion active**

Parameters:	%1 = Channel number
	%2 = Axis name, spindle number
Definitions:	'Fixed-point approach in JOG' is active for the axis, but another offset motion - for example a synchronization offset $AA_OFF - has been interpolated simultaneously.
	The position of the selected fixed-point is not reached if offset values are changed during the traversing motion.
	The target point then becomes "fixed-point position + change in offset value".
	The end point will be reached if the traversing motion is restarted after the offset value has been changed.
	(For example: incremental traversing in which the traversing motion stops intermittently).
	Reason:
	Restarting the motion takes the current offset value into account.
Reaction:	Interface signals are set.
	Alarm display.
Remedy:	Trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17814   **[Channel %1:] Axis %2 fixed-point position not available**

Parameters:	%1 = Channel number
	%2 = Axis name, spindle number
	%3 = Number of fixed-point position

Definitions:	No fixed-point position is available for the fixed point selected in JOG mode. See MD30610 $MA_NUM_FIX_POINT_POS.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Adapt MD30610 $MA_NUM_FIX_POINT_POS and, if necessary, MD30600 $MA_FIX_POINT_POS[]. Deselect fixed-point approach or select a valid fixed point, and restart the JOG motion.
Program Continuation:	Clear alarm with the Delete key or NC START.

17815 Indexing axis %1 fixed point %2 unequal indexing position

Parameters:	%1 = Axis number %2 = Array index of machine data
Definitions:	The axis is a referenced indexing axis, and the fixed-point number %2 to be approached in JOG mode (defined in MD30600 $MA_FIX_POINT_POS) does not coincide with an indexing position. In JOG mode, referenced indexing axes approach indexing positions.
Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	MD30600 $MA_FIX_POINT_POS[] or adapt the indexing positions.
Program Continuation:	Clear alarm with the RESET key. Restart part program

17820 [Channel %1:] JOG to position not possible for axis %2, reason %3

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Cause
Definitions:	A 'JOG to position' has been requested for an axis. This is not possible because: Reason 1: The axis is involved in the active transformation. Reason 2: The axis is a following axis in an active coupling. The JOG to position will therefore not be executed.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Deselect 'JOG to position', or previously deselect the transformation with TRAFOOF or disband the coupling.
Program Continuation:	Clear alarm with the Delete key or NC START.

17821 [Channel %1:] Axis %2 JOG to position and override motion active

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	'JOG to position' is active for the axis, but an offset motion - for example a synchronization offset $AA_OFF - has been interpolated simultaneously. The position of the SD43320 $SA_JOG_POSITION is not reached if offset values are changed during the traversing motion. The target point then becomes "Jog position + change in offset value". The position SD43320 $SA_JOG_POSITION will be reached if the traversing motion is restarted after the offset value has been changed. (For example: incremental traversing in which the traversing motion stops intermittently). Reason: Restarting the motion takes the current offset value into account.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17822	**[Channel %1:] Axis %2 JOG to position: Position changed**
Parameters:	%1 = Channel number
Definitions:	An axis motion is active for the axis with 'JOG to position' but the position, that is the content of SD43320 $SA_JOG_POSITION, has been changed. The approach motion is canceled.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17823	**[Channel %1:] Axis %2 JOG to position deactivated**
Parameters:	%1 = Channel number
Definitions:	An axis motion is active for the axis with 'JOG to position' but 'JOG to position' has been deactivated. The approach motion is canceled.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17825	**Indexing axis %1 $SA_JOG_POSITION unequal indexing position**
Parameters:	%1 = Axis number
Definitions:	The axis is a referenced indexing axis and 'JOG to position' is activated in JOG mode, but SD43320 $SA_JOG_POSITION does not coincide with an indexing position. In JOG mode, referenced indexing axes approach indexing positions.
Reaction:	NC not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify SD43320 $SA_JOG_POSITION or indexing positions.
Program Continuation:	Clear alarm with the RESET key. Restart part program

17830	**[Channel %1:] JOG in a circle is activated, but the axis %2 required for this is not a geometry axis.**
Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The function JOG in circles has been activated, but the axis required for this has not been defined as a geometry axis.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Define axis as geometry axis.
Program Continuation:	Clear alarm with the Delete key or NC START.

17831	**[Channel %1:] JOG a circle is not possible, reason %2**
Parameters:	%1 = Channel number %2 = Cause
Definitions:	The JOG in circles was activated, but this is not possible because: 1. The current positions of the axes involved lie outside the selected pitch circle. 2. The current positions of the axes involved, with pitch circle selected and tool radius offset active, are too near to the center of the circle. 3. The current positions of the axes involved, with tool radius offset active, are too near to the limiting circle during internal machining.

4. The current positions of the axes involved, with tool radius offset active, are too near to the limiting circle during external machining.

5. The current positions of the axes involved in internal machining are outside the defined circle.

6. The current positions of the axes involved in external machining are inside the defined circle.

10. A rotation is acting on the current plane, that is the current plane is inclined in space. This is not currently supported.

Reaction:	Interface signals are set. Alarm display.
Remedy:	Define axis as geometry axis.
Program Continuation:	Clear alarm with the Delete key or NC START.

17833 [Channel %1:] JOG a circle is active and JOG circles deactivated

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	A circular motion is active but 'JOG in circles' has been deactivated. The circular motion is canceled.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Reactivate 'JOG circles' and trigger JOG motion again.
Program Continuation:	Clear alarm with the Delete key or NC START.

17900 [Channel %1:] Block %2 motion synchronous action: %3 axis %4 is not a machine axis

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Axis name
Definitions:	At this point, the block context calls for a machine axis. This is the case with: - G74 (reference point approach) - G75 (fixed point approach) If a geometry or additional axis identifier is used, then it must also be allowed as machine axis identifier (MD10000 $MN_AXCONF_MACHAX_NAME_TAB).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Use machine axis identifier when programming.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18000 [Channel %1:] Block %2 NCK-specific protection zone %3 wrong. Error code %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of NCK protection zone %4 = Error specification
Definitions:	There is an error in the definition of the protection zone. The error number gives the specific reason for the alarm: No.Meaning 1: Incomplete or conflicting contour definition. 2: Contour encompasses more than one surface area. 3: Tool-related protection zone is not convex. 4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value. 5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones). 6: Protection zone definition consists of more than 10 contour elements. 7: Tool-related protection zone is defined as inside protection zone. 8: Incorrect parameter used. 9: Protection zone to be activated is not defined.

10: Incorrect modal G code used for protection zone definition.
11: Contour definition incorrect or frame activated.
12: Other, not further specified errors.

Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify definition of the protection zone and check MD.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18001   [Channel %1:] Block %2 channel-specific protection zone %3 incorrect. Error code %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of the channel-specific protection zone %4 = Error specification
Definitions:	There is an error in the definition of the protection zone. The error number gives the specific reason for the alarm. No.Meaning 1: Incomplete or conflicting contour definition. 2: Contour encompasses more than one surface area. 3: Tool-related protection zone is not convex. 4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value. 5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones). 6: Protection zone definition consists of more than 10 contour elements. 7: Tool-related protection zone is defined as inside protection zone. 8: Incorrect parameter used. 9: Protection zone to be activated is not defined. 10: Incorrect modal G code used for protection zone definition. 11: Contour definition incorrect or frame activated. 12: Other, not further specified errors.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify definition of the protection zone and check MD.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18002   [Channel %1:] Block %2 NCK protection zone %3 cannot be activated. Error code %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of NCK protection zone %4 = Error specification
Definitions:	An error has occurred on activating the protection zone. The error number gives the specific reason for the alarm. No.Meaning 1: Incomplete or conflicting contour definition. 2: Contour encompasses more than one surface area. 3: Tool-related protection zone is not convex. 4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value. 5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones). 6: Protection zone definition consists of more than 10 contour elements. 7: Tool-related protection zone is defined as inside protection zone. 8: Incorrect parameter used. 9: Protection zone to be activated is not defined or number of contour element <2 or >MAXNUM_CONTOURNO_PROTECTAREA. 10: Error in internal structure of the protection zones. 11: Other, not further specified errors. 12: The number of protection zones simultaneously active exceeds the maximum number (channel-specific machine data).

13,14: Contour element for protection zones cannot be created.
15,16: No more memory space for the protection zones.
17: No more memory space for the contour elements.

Reaction:	Correction block is reorganized.
	Interface signals are set.
	Alarm display.
	If the alarm is output on ramp-up (2nd parameter: "INIT" instead of block number), "Channel not ready to operate" will be set.

Remedy: Please inform the authorized personnel/service department.
1. Reduce the number of simultaneously active protection zones (MD).
2. Modify part program:
- Delete other protection zones.
- Preprocessing stop.
When the alarm occurs during control ramp-up, the system variables $SN_PA_... have to be corrected for the specified protection zone. Afterwards perform a restart. If the erroneous data cannot be recognized, the protection zone's immediate activation can be removed, and the system variables of the protection zone can be written again by means of NPROTDEF.

Program Continuation: Clear alarm with NC START or RESET key and continue the program.
If the alarm occurs during NC program execution, the current block can be changed. This way, the NPROT parameters can also be adjusted. However, if there is an error in the definition of the protection zone, the NC program must be aborted and the definition must be corrected under NPROTDEF.
If the alarm occurs on control ramp-up, system variables $SN_PA_... must be corrected for the specified protection zone. This can be done by downloading an Initial.ini file that includes the relevant corrected date. If afterwards a restart is performed again, the alarm will have been removed provided that the data are consistent.

18003 [Channel %1:] Block %2 channel-specific protection zone %3 cannot be activated. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of the channel-specific protection zone
%4 = Error specification

Definitions: An error has occurred on activating the protection zone. The error number gives the specific reason for the alarm.
No.Meaning
1: Incomplete or conflicting contour definition.
2: Contour encompasses more than one surface area.
3: Tool-related protection zone is not convex.
4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
5: The number of the protection zone does not exist (negative number, zero or greater than the maximum number of protection zones).
6: Protection zone definition consists of more than 10 contour elements.
7: Tool-related protection zone is defined as inside protection zone.
8: Incorrect parameter used.
9: Protection zone to be activated is not defined or number of the contour element <2 or >MAXNUM_CONTOURNO_PROTECTAREA.
10: Error in internal structure of the protection zones.
11: Other, not further specified errors.
12: The number of protection zones simultaneously active exceeds the maximum number (channel-specific machine data).
13,14: Contour element for protection zones cannot be created.
15,16: No more memory space for the protection zones.
17: No more memory space for the contour elements.

Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
If the alarm is output on ramp-up (2nd parameter: "INIT" instead of block number), "Channel not ready to operate" will be set.

Remedy:	Please inform authorized personnel / the service department. 1. Reduce the number of simultaneously active protection zones (MD). 2. Modify part program: - Delete other protection zones. - Preprocessing stop. When the alarm occurs on control ramp-up, system variables \$SC_PA_... must be corrected for the specified protection zone. Afterwards perform a restart. If the erroneous data cannot be recognized, the protection zone's immediate activation can be removed, and the system variables of the protection zone can be written again by means of CPROTDEF.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program. The current block can be changed if the alarm occurs during NC program execution. The CPROT parameters can also be adjusted. However, if the error lies in the definition of the protection zone, the NC program must be aborted and the definition corrected under CPROTDEF. If the alarm occurs on control power-up, the system variables \$SC_PA_... must be corrected for the specified protection zone. This can be done by downloading an Initial.ini file that includes the relevant corrected data. If another restart is then made, the alarm will have been eliminated provided that the data are now consistent.

18004 [Channel %1:] Block %2 orientation of workpiece-related protection zone %3 does not correspond to the orientation of tool-related protection zone %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Number of workpiece-related protection zone
Definitions:	The orientation of the workpiece-related protection zone and the orientation of the tool-related protection zone differ. If the protection zone number is negative, then this is an NCK protection zone.
Reaction:	Correction block is reorganized. Interface signals are set. Alarm display.
Remedy:	- Modify the protection zone definition or do not simultaneously activate protection zones that have different orientations. - Check machine data and modify the protection zone definition if necessary.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18005 [Channel %1:] Block %2 serious error in definition of NCK-specific protection zone %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Protection zone number
Definitions:	The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed. This also applies to any that are initiated implicitly such as with G74, M30, M17.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18006 [Channel %1:] Block %2 serious error in definition of channel-specific protection zone %3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Protection zone number
Definitions:	The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed. This also applies to any that are initiated implicitly such as with G74, M30, M17.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set.

	Alarm display.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18205 [Channel %1:] Block %2 motion synchronous action: %3 curve table %4 does not exist

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Number of curve table
Definitions:	An attempt was made to use a curve table whose table number is not known in the system \par.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Change the table number in the program instruction or define the curve table with the desired table number.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18300 [Channel %1:] Block %2 frame: fine shift not possible

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Allocation of a fine shift to settable frames or the basic frame is not possible since MD18600 $MN_MM_FRAME_FINE_TRANS is not equal to 1.
Reaction:	Interpreter stop Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Modify program or set MD18600 $MN_MM_FRAME_FINE_TRANS to 1.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

18310 [Channel %1:] Block %2 frame: illegal rotation

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Rotations are not possible with NCU global frames.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18311 [Channel %1:] Block %2 frame: illegal instruction

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	An attempt was made to read or write a frame which does not exist.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.

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

---

18312 [Channel %1:] Block %2 frame: fine shift not configured

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	Fine shift must be configurd with G58 and G59.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify machine data.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18313 [Channel %1:] Block %2 frame: illegal switchover of geometry axes

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	It is not allowed to change the geometry axis assignment because the current frame contains rotations.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Change NC program or set other mode with MD10602 $MN_FRAME_GEOAX_CHANGE_MODE.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18314 [Channel %1:] Block %2 frame: type conflict

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	It is not possible to chain global frames and channel-specific frames. The alarm occurs if a global frame is programmed with a channel axis name and no machine axis on this NCU is assigned to the channel axis. Channel-specific frames cannot be programmed with machine axis names if there is no corresponding channel axis on this NCU.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

18400 [Channel %1:] Block %2 language change not possible:%3

Parameters:	%1 = Channel number %2 = Block number, label %3 = Cause
Definitions:	The selection of an external NC language is not possible due to the reason specified. The following reasons are possible (see parameter 3): 1. Invalid machine data settings 2. Active transformation
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.

Remedy:	Remedy the specified cause of the error before selecting the language.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20000 [Channel %1:] Axis %2 reference cam not reached

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	After starting the reference point approach, the rising edge of the reduction cam must be reached within the section defined in the MD34030 $MA_REFP_MAX_CAM_DIST (phase 1 of referencing). (This error occurs only with incremental encoders).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. There are 3 possible causes of error: 1. The value entered in MD34030 $MA_REFP_MAX_CAM_DIST is too small. Determine the maximum possible distance from the beginning of reference motion up to the reduction cam and compare with the value in MD34030 $MA_REFP_MAX_CAM_DIST, increase the value in the MD if necessary. 2. The cam signal is not received by the PLC input module. Operate the reference point switch manually and check the input signal on the NC/PLC interface (route: Switch! Connector! Cable! PLC input! User program). 3. The reference point switch is not operated by the cam. Check the vertical distance between reduction cam and activating switch.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20001 [Channel %1:] Axis %2 no cam signal present

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	At the beginning of phase 2 of reference point approach, the signal from the reduction cam is no longer available. Phase 2 of reference point approach begins when the axis remains stationary after deceleration to the reduction cam. The axis then starts in the opposite direction in order to select the next zero marker of the measuring system on leaving the reduction cam or approaching it again (negative/positive edge).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check whether the deceleration path after the approach velocity is greater than the distance to the reference point cam - in which case the axis cannot stop until it is beyond the cam. Use a longer cam or reduce the approach velocity in MD34020 $MA_REFP_VELO_SEARCH_CAM. When the axis has stopped at the cam, it must be checked whether the signal DB380x DBX1000.7 (Deceleration reference point approach) is still available at the interface to the NCK. - Hardware: Wire break? Short circuit? - Software: User program?
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20002 [Channel %1:] Axis %2 zero mark not found

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The hardware zero mark of the incremental position encoder or the substitute zero mark of the absolute position encoder is not within a defined section. Phase 2 of the reference point approach ends when the zero mark of the encoder has been detected after the rising/falling edge of the NC/PLC interface signal DB380x DBX1000.7 (Deceleration reference point approach) has given the trigger start. The maximum distance between the trigger start and the zero mark that follows is defined in the MD34060 $MA_REFP_MAX_MARKER_DIST.

	The monitor prevents a zero mark signal from being overtraveled and the next being evaluated as reference point signal. (Faulty cam adjustment or excessive delay by the PLC user program).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check the cam adjustment and make sure that the distance is sufficient between the end of the cam and the zero marker signal that follows. The path must be greater than the axis can cover in the PLC cycle time. Increase the MD34060 $MA_REFP_MAX_MARKER_DIST, but do not select a value greater than the distance between the 2 zero markers. This might result in the monitor being switched off.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20003 [Channel %1:] Axis %2 measuring system error

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	In a measuring system with distance-coded reference marks, the distance between two adjacent markers has been found to be more than twice the value entered in MD34300 $MA_ENC_REFP_MARKER_DIST. The control does not issue the alarm until it has again detected a distance that is too long after having made a 2nd attempt in reverse direction with half the traversing velocity.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Determine the distance between 2 odd reference marks (reference mark interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in MD34060 $MA_REFP_MAX_MARKER_DIST. Check the reference track of the scale including the electronics for the evaluation.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20004 [Channel %1:] Axis %2 reference mark missing

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	In the distance-coded length measurement system, 2 reference marks were not found within the defined searching distance (axis-specific MD34060 $MA_REFP_MAX_MARKER_DIST). A reduction cam is not required for distance-coded scales (but an existing cam will be evaluated). The conventional direction key determines the direction of search. The searching distance MD34060 $MA_REFP_MAX_MARKER_DIST, within which the two reference marks are expected is counted commencing at the start point.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Determine the distance between 2 odd reference point markers (reference point marker interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in the MD34060 $MA_REFP_MAX_MARKER_DIST. Check the reference point track of the scale including the electronics for the evaluation.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20005 [Channel %1:] Axis %2 reference point approach aborted

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	Referencing could not be completed for all stated axes (e.g., abort caused by missing servo enable, measuring system switchover, release of direction key, etc.).

In distance-coded measuring systems, the alarm will also be displayed if the value 1 has been set in MD34000 $MA_REFP_CAM_IS_ACTIV (reference cams) and one of the conditions stated in the remedy has been fulfilled.

Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check the possible reasons for termination: - Servo enable missing: NC/PLC interface signal DB380x DBX2.1 (Servo enable) - Measuring system switchover: NC/PLC interface signal DB380x DBX1.5 / 1.6 (Position measuring system 1/2) - Traversing key + or - missing: NC/PLC interface signal DB380x DBX4.7 / 4.6 (Traversing keys plus/minus) - Feed override = 0 - The feed disable is active - Exact stop not reached within MD36020 $MA_POSITIONING_TIME. The axis-specific MD34110 $MA_REFP_CYCLE_NR determines which axes are involved in the channel-specific referencing. ValueMeaning -1: No channel-specific referencing, NC Start without referencing. 0: No channel-specific referencing, NC Start with referencing. 1-8: Channel-specific referencing. The number entered here corresponds to the referencing sequence. (When all axes with contents 1 have reached the reference point, then the axes with contents 2 start, etc.).
Program Continuation:	Clear alarm with the RESET key. Restart part program

20006 [Channel %1:] Axis %2 reference point creep velocity not reached

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	In phase 2 of reference point approach (wait for zero mark), the cam end was reached but the reference point approach velocity was not within the tolerance window. (This can occur when the axis is already at the end of the cam at the beginning of reference point approach. Phase 1 is therefore considered as being already concluded and will not be started.) Phase 2 has been interrupted (this time before the cam) and the reference point approach will be started once again automatically with phase 1. If the approach velocity is not reached at the 2nd attempt either, referencing will be stopped and the alarm displayed. Approach velocity: MD34040 $MA_REFP_VELO_SEARCH_MARKER Velocity tolerance: MD35150 $MA_SPIND_DES_VELO_TOL.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Reduce the MD for the approach velocity MD34040 $MA_REFP_VELO_SEARCH_MARKER and/or increase the MD for the velocity tolerance MD35150 $MA_SPIND_DES_VELO_TOL.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20007 [Channel %1:] Axis %2 reference point approach requires 2 measuring systems

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	2 encoders are needed for setting MD34200 $MA_ENC_REFP_MODE = 6!
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Modify reference mode MD34200 $MA_ENC_REFP_MODE or install and configure a second encoder.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20008 **[Channel %1:] Axis %2 reference point approach requires second referenced measuring system**

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	When setting MD34200 $MA_ENC_REFP_MODE = 6 the 2nd encoder must first be referenced.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify referencing mode MD34200 $MA_ENC_REFP_MODE or reference 2nd encoder.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20050 **[Channel %1:] Axis %2 handwheel mode active**

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The axes cannot be traversed in JOG mode using the traversing keys because traversing is still taking place via the handwheel.
Reaction:	Alarm display.
Remedy:	Decide whether the axis is to be traversed by means of the direction keys or the handwheel. End handwheel travel and delete the axial distance-to-go if necessary (NC/PLC interface signal DB380x DBX2.2 (Delete distance-to-go/ Spindle reset)).
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20051 **[Channel %1:] Axis %2 handwheel mode not possible**

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The axis is already traveling via the traversing keys, so handwheel mode is no longer possible.
Reaction:	Alarm display.
Remedy:	Decide whether the axis is to be traversed by means of the jog keys or via the handwheel.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20052 **[Channel %1:] Axis %2 already active**

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The axis is to traverse as a machine axis in JOG mode using the direction keys on the machine control panel. However, this is not possible because: 1. It is already traversing as a geometry axis (through the channel-specific interface DB3200 DBX1000.7 / 0.6 (Traversing keys -/+) or DB3200 DBX1004.7 / 4.6 (Traversing keys -/+) or DB3200 DBX1008.7 / 8.6 (Traversing keys -/+)) or 2. It is already traversing as a machine axis (through the axis-specific interface DB380x DBX4.7 / 4.6 (Traversing keys plus/minus)) or 3. A frame is valid for a rotated coordinate system, and another geometry axis involved in this is already traversing in JOG mode by means of the direction keys.
Reaction:	Alarm display.
Remedy:	Stop traversing through the channel or axis interface or stop the other geometry axis.
Program Continuation:	Clear alarm with the Delete key or NC START.

20053 **[Channel %1:] Axis %2 DRF, FTOCON, external zero point offset not possible**

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

Definitions:	The axis is traversed in a mode (e.g. referencing) that allows no additional overlaid interpolation.
Reaction:	Alarm display.
Remedy:	Wait until the axis has reached its reference position or terminate reference point approach with "Reset" and start DRF once again.
Program Continuation:	Clear alarm with the Delete key or NC START.

20054   [Channel %1:] Axis %2 wrong index for indexing axis in JOG mode

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	1. The displayed indexing axis is to be traversed incrementally in JOG mode (by 1 indexing position). However, no further indexing position is available in the selected direction. 2. The axis is stationary at the last indexing position. In incremental traversing the working area limitation or the software limit switch is reached without an indexing position being located in front of it at which a stop could be made.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Correct (add to) the list of indexing positions by means of the machine data MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1 MD10910 $MN_INDEX_AX_POS_TAB_1 MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2 MD10930 $MN_INDEX_AX_POS_TAB_2 or set the working area limits or the software limit switches to other values.
Program Continuation:	Clear alarm with the Delete key or NC START.

20055   [Channel %1:] Master spindle not present in JOG mode

Parameters:	%1 = Channel number
Definitions:	The displayed axis is to be traversed as machine axis in JOG mode with revolutional feed, but no master spindle has been defined from which the actual speed could have been derived.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. If the revolutional feed is also to be active in JOG mode, then a master spindle must be declared via the channel-specific MD20090 $MC_SPIND_DEF_MASTER_SPIND. In this case you have to open a screen in the PARAMETER operating area with the softkeys "SETTINGDATA" and "JOG DATA" and preselect the G function G95 there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feed, the control takes the value assigned in the axis-specific MD 32050 $MA_JOG_REV_VELO or in the case of rapid traverse overlay MD32040 $MA_JOG_REV_VELO_RAPID). The revolutional feed in JOG mode is deactivated by changing the G function from G95 to G94.
Program Continuation:	Clear alarm with the Delete key or NC START.

20056   [Channel %1:] Axis %2 no revolutional feedrate possible. Axis/spindle %3 stationary

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Axis name, spindle number
Definitions:	An axis is to travel in JOG with revolutional feed, but the spindle/axis the feed is to be derived from is 0.
Reaction:	Alarm display.
Remedy:	Traverse the spindle/axis from which the feed is to be derived.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20057 **[Channel %1:] Block %2 revolutional feedrate for axis/spindle %3 is <= zero**

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number
Definitions:	Revolutional feed has been programmed for an axis/spindle, but the velocity was not programmed or the programmed value is smaller than or equal to zero.
Reaction:	Correction block is reorganized. Local alarm reaction. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. - Correct the part program or - Specify the correct feed for PLC axes at the VDI interface, - Specify feed for oscillating axes in the SD43740 $SA_OSCILL_VELO.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20058 **[Channel %1:] Axis %2 revolutional feedrate: illegal feed source**

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	An axis/spindle is to be traversed at revolutional feedrate. The reference axis/spindle defined in SD 43300 $SA_ASSIGN_FEED_PER_REV_SOURCE refers to itself. The coupling caused cannot be executed.
Reaction:	Alarm display.
Remedy:	The reference axis/spindle must be modified accordingly in SD 43300.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20059 **[Channel %1:] Axis %2 already active due to %3**

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Cause
Definitions:	The axis (machine axis, geometry axis or orientation axis) is to be traversed in operation mode "Automatic&Jog" (see MD10735 $MN_JOG_MODE_MASK) by using the direction keys or a handwheel. This is not possible, as (see parameter 3): 1. the axis is active as a rotating spindle 2. the axis is a PLC axis 3. the axis is active as an asynchronous reciprocating axis 4. the axis is active as a command axis 5. the axis is active as a slave axis 6. a frame applies for a rotated coordinate system and an axis involved in the required JOG movement of the geometry axis is not available for this 7. an axis container rotation is activated via NCU link Note: This alarm identifies an axis not capable of JOG which received a JOG order. In this case, the NCK will not proceed according to "Internal JOG".
Reaction:	Alarm display.
Remedy:	Wait for the axis to traverse or abort with distance-to-go delete or RESET.
Program Continuation:	Clear alarm with the Delete key or NC START.

20060 **[Channel %1:] Axis %2 cannot be traversed as geometry axis**

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

Definitions:	The axis is currently not in "Geometry axis" state. Therefore, it cannot be traversed in JOG mode as geometry axis. If the abbreviation work (workpiece coordinate system) is displayed in the "Position" screen, then only the geometry axes can be traversed by means of the direction keys! (MCS ... Machine coordinate system; all machine axes can now be traversed by using the direction keys on the machine control panel).
Reaction:	Alarm display.
Remedy:	Check the operating steps to establish whether geometry axes really must be traversed, otherwise switch over to the machine axes by activating the "Work/Machine" key on the machine control panel.
Program Continuation:	Clear alarm with the Delete key or NC START.

20062 [Channel %1:] Axis %2 already active

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The displayed axis is already traversing as a machine axis. Therefore, it cannot be operated as a geometry axis. Traversing an axis can take place in JOG mode through 2 different interfaces. 1. As a geometry axis: via the channel-specific interface DB3200 DBX1000.7 / 0.6 (Traversing keys -/+) 2. As a machine axis: via the axis-specific interface DB380x DBX4.7 / 4.6 (Traversing keys plus/minus) With the standard machine control panel, it is not possible to operate an axis as a machine axis and as a geometry axis at the same time.
Reaction:	Alarm display.
Remedy:	Do not start the geometry axis until the traversing motion as machine axis has been concluded.
Program Continuation:	Clear alarm with the Delete key or NC START.

20064 [Channel %1:] Axis %2 selection of several axes with an active taper angle is not permitted.

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	With an active taper angle, only one geometry axis at the time can be traversed in JOG mode by pressing traversing keys. Simultaneous traversing of a geometry axis as a machine axis is not permitted either.
Reaction:	NC not ready. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Starting the geometry axis only if traversing of the other geometry axis or machine axis completed.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20065 [Channel %1:] Master spindle not defined for geometry axes in JOG mode

Parameters:	%1 = Channel number
Definitions:	The displayed axis is to be traversed as geometry axis in JOG mode with rotary feed, but no master spindle has been defined from which the actual speed could be derived.
Reaction:	Local alarm reaction. Interface signals are set. Alarm display.
Remedy:	If the revolutional feedrate is to be active in JOG mode too, then a master spindle must be declared in the channel-specific machine data MD20090 $MC_SPIND_DEF_MASTER_SPIND. In this case, you have to open a screen in the PARAMETER operating area with the softkeys "SETTINGDATA" and "JOG DATA", and preselect the G function G95 there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feedrate, the control takes the value assigned in the axis-specific machine data MD32050 $MA_JOG_REV_VELO or in the case of rapid traverse override MD32040 $MA_JOG_REV_VELO_RAPID). The revolutional feedrate in JOG mode is deactivated by changing the G function from G95 to G94.
Program Continuation:	Clear alarm with the Delete key or NC START.

20080 [Channel %1:] Axis %2 no handwheel assigned for override

Parameters:	%1 = Channel number
	%2 = Axis number
Definitions:	No handwheel has been assigned for this specified axis after handwheel overlay has been started in automatic mode. If the axis identifier is missing in the alarm with active velocity overlay FD > 0, then the 1st geometry axis has not been defined in the NC channel. In this case the block is executed without handwheel control.
Reaction:	Alarm display.
Remedy:	If handwheel control is required, a handwheel must be activated.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20085 [Channel %1:] Contour handwheel: traverse direction or overtravel of beginning of block not allowed

Parameters:	%1 = Channel number
Definitions:	Travel takes place on the path with the contour handwheel in the opposite direction to the programmed travel direction and the starting point of the path has been reached at the start of the block.
Reaction:	Alarm display.
Remedy:	Turn the contour handwheel in the opposite direction.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

20138 [Channel %1:] Block %2 motion synchronous action %3 command axis %4 cannot be traversed

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
	%4 = Axis name
Definitions:	An NC alarm was detected for a command axis which is to be traversed from a synchronous action.
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20139 [Channel %1:] Block %2 motion synchronous action: %3 invalid marker

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
Definitions:	Setting or deleting of a marker in the motion-synchronous action is not possible.
	Possible causes:
	SETM(): Maximum number of markers exceeded; marker has already been set.
	CLEARM(): Specified marker is not within permissible value range.
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	SETM(): use marker in valid value range; do not set the marker again.
	CLEARM(): use marker in valid value range.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20140   **[Channel %1:] Traversing of command axis %2 see NC alarm %3 parameter %4**

Parameters:	%1 = Channel number %2 = Axis %3 = NC alarm %4 = Additional parameter
Definitions:	An NC alarm was detected for a command axis which is to be traversed from a synchronized action. The NC alarm is indicated by an MMC alarm number in the 3rd parameter. If there is any additional information, this will be provided in a 4th parameter.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	See help information for the additional alarms.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20144   **[Channel %1:] Block %2 motion synchronous action: %3 system variable cannot be accessed**

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	When using system variables, it is assumed that a read/write operation can access the required data successfully. In accesses to encoder actual values or digital I/Os, the result depends on the availability of the corresponding hardware components. If an access within synchronized actions does not return a valid value, alarm 20144 is output. Outside synchronized actions, such a read/write access causes block execution to be interrupted until the result is available. Block execution is subsequently continued.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Before reading/writing system variables, ensure that it is possible to access the required hardware components.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20145   **[Channel %1:] Block %2 motion synchronous action: %3 arithmetic error**

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	In calculating an arithmetic expression for a motion synchronous action, an overflow has occurred (e.g. division by zero).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Correct error in expression.
Program Continuation:	Clear alarm with the RESET key. Restart part program

20146   **[Channel %1:] Block %2 motion synchronous action: %3 nesting depth exceeded**

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	For calculating arithmetic expressions in motion synchronous blocks, an operand stack with a fixed set size is used. With very complex expressions, this stack can overflow.

Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Correct error in expression.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20147   **[Channel %1:] Block %2 motion synchronous action: %3 command %4 not executable**

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
	%4 = Program command
Definitions:	One of the commands for the synchronous action block cannot be executed, e.g. it is not possible to perform a Reset to the synchronous action.
	Measurement level 2
	- Embargo version does not allow measurement from a synchronized action
	- MEASA was programmed in a synchronized action
	- Measurement is already active
	- Programming error (see alarm 21701)
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Change synchronous action.
	Measurement level 2
	Execute the measurement task from an NC program first, in order to improve the error diagnostics. Only include it in the synchronized action when the first error-free run has been performed.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20148   **[Channel %1:] Block %2 motion synchronous action: %3 internal error %4**

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
	%4 = Error code
Definitions:	An internal error has occurred during processing of a synchronous action. The error code is for diagnostics purposes. Please make a note and contact the manufacturer.
Reaction:	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Change synchronous action.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20149   **[Channel %1:] Block %2 motion-synchronous action: %3 Index %4 is illegal**

Parameters:	%1 = Channel number
	%2 = Block number, line number
	%3 = Synact ID
	%4 = Index
Definitions:	An illegal index was used to access a variable in the motion-synchronous action. The illegal index is displayed.
	Example: ... DO $R[$AC_MARKER[1]] = 100
	The error occurs if the value of marker 1 is greater than the maximum permissible R-parameter number.

Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Use a valid index.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

20170   [Channel %1:] Machine data $AC_FIFO invalid

Parameters:	%1 = Channel number
Definitions:	the structure of the FIFO variable $AC_FIFO1 - $AC_FIFO10 determined by machine data MD28260 $MC_NUM_AC_FIFO, MD28262 $MC_START_AC_FIFO, MD28264 $MC_LEN_AC_FIFO, MD28266 $MC_MODE_AC_FIFO cannot be stored in the R parameter field defined in MD28050 $MC_MM_NUM_R_PARAM.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Increase the number of the R parameters or reduce the FIFO elements. MD28050 $MC_MM_NUM_R_PARAM = MD28262 $MC_START_AC_FIFO + MD28260 $MC_NUM_AC_FIFO * (MD28264 $MC_LEN_AC_FIFO + 6)
Program Continuation:	Switch control OFF - ON.

20205   [Channel %1:] Block %2 motion synchronous action: %3 invalid spindle number %4

Parameters:	%1 = Channel number target channel %2 = Block number, line number %3 = Synact ID %4 = Spindle number
Definitions:	There is no spindle/axis assignment in the target channel for the specified spindle.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21550   [Channel %1:] Axis %2 Travel from hardware limit switch not possible. Reason: %3

Parameters:	%1 = Channel number %2 = Axis name %3 = Cause
Definitions:	It has been tried to retract a following axis of an axis coupling or an output axis of a transformation through the master axis or input axis of a transformation. This is not permitted in the current situation. Possible reasons: 1 No permissible direction of retraction 2 Coupling not synchronous 3 Retraction not permitted for the active coupling 4 Reserved 5 Retraction not permitted for the active transformation
Reaction:	NC Start disable in this channel. Alarm display.
Remedy:	Remedy for error cause: 1 Define another travel direction 2 Deactivate the coupling and travel the axis/axes separately 3 Deactivate the coupling and travel the axis/axes separately 4 Reserved 5 Deactivate the transformation and travel the axis/axes separately

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

---

21610   [Channel %1:] Axis %2 encoder %3 frequency exceeded

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = String (encoder number)
Definitions:	The maximum permissible frequency of the currently active encoder (axis-specific interface signal DB380x DBX1.5 / 1.6 (position measuring system 1/2)) in the axis-specific MD36300 $MA_ENC_FREQ_LIMIT [n] (n ... encoder number, 1 or 2) has been exceeded. The reference of the actual value to the mechanical slide position may have been lost. The alarm can be reprogrammed in MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Check MD36300 $MA_ENC_FREQ_LIMIT [n] and NC/PLC interface signal DB380x DBX1.5 / 1.6 (position measuring system 1/2).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

21612   [Channel %1:] Axis %2: enable reset, cause %3

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Cause of the alarm

Definitions:	Causes of alarm:
	  0: The cause of the alarm cannot be precisely determined.
	  1: The interface signal DB380x DBX2.1 (Servo enable) is missing
	  2: The interface signal DB380x DBX4001.7 (Pulse enable) is missing
	  3: Drive signal DB390x DBX4001.7 (Impulses enabled) is not set
	  4: Drive signal DB390x DBX4001.5 (Drive ready) is not set
	  5: Drive signal DB390x DBX4000.4 (Autonomous drive) does not follow the NC setpoints
	One of the motion-enabling signals (e.g. "Servo enable", "Pulse enable", parking/encoder selection (only for axes) or drive-specific enables (such as terminal 663 with SIMODRIVE 611D) has been reset for the displayed axis. The alarm can be reported with positioning axes, spindles and for axes from the geometry grouping.
	The axes entered in the channel-specific MD array MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB are regarded as axes belonging to the geometry grouping. Servo enable must exist for all available geometry axes, regardless of whether or not they are currently in motion.
	Occurs in connection with SAFETY function: If a test stop is performed with linked axes, the alarm is issued if a motion command from the ELG grouping is pending during the test stop of the following axis.

Reaction:	The NC switches to follow-up mode. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check the interface signals DB380x DBX2.1 (Servo enable), DB380x DBX4001.7 (Pulse enable), check the drive signals DB390x DBX4001.7 (Pulses enabled), DB390x DBX4001.5 (Drive ready) for example with the PLC status display in the DIAGNOSTICS operating area. Check the encoder selection (for axes) as well as other signals enabling motion (such as SIMODRIVE 611D terminal 663 etc.) according to the drive type used. When the terminal enables of the drive have failed, trace back the wiring or hardware function (for example relay function) or proceed as stated in the relevant drive documentation. With SAFETY: With active actual-value linkage, output of the error message on the slave axis can be prevented by increasing MD36060 $MA_STANDSTILL_VELO_TOL (default value is 5 mm).
Program Continuation:	Clear alarm with the Delete key or NC START.

21613	**Axis %1 measuring system changing**
Parameters:	%1 = Axis name, spindle number
Definitions:	The measuring system for this axis is changing.
Reaction:	Alarm display.
Remedy:	-
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

21614	**[Channel %1:] Axis %2 hardware limit switch %3**
Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = String (+, - or +/-)
Definitions:	The signal DB380x DBX1000.1 und .0 (Hardware limit switch plus/minus) has been set at the NC/PLC interface.
Reaction:	NC Start disable in this channel. Alarm display.
Remedy:	Please inform the authorized personnel/service department. 1. With axes that have already been referenced, the software limit switch 1 or 2 should respond before the hardware limit switch is reached. Check MD36110 $MA_POS_LIMIT_PLUS, MD36100 $MA_POS_LIMIT_MINUS, MD36130 $MA_POS_LIMIT_PLUS2 and MD36120 $MA_POS_LIMIT_MINUS2 and the NC/PLC interface signal for the selection DB380x DBX1000.3 / 1000.2 (1st/2nd software limit switch plus/minus) and correct, if necessary (PLC user program). 2. If the axis has not yet been referenced, it is possible to depart from the hardware limit switch in the opposite direction in JOG mode. 3. Check the PLC user program and the connection from the switch to the PLC input module, provided the axis has not reached the hardware limit switch at all.
Program Continuation:	Clear alarm with the RESET key. Restart part program

21615	**[Channel %1:] Axis %2 taken from traverse mode to follow-up mode**
Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	This axis has been taken from traverse mode and put into "Follow-up" mode, for instance because the pulse enable for the drive has been reset.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	-
Program Continuation:	Clear alarm with the RESET key. Restart part program

21620	**[Channel %1:] Axis %2 Emergency braking ramp activated**
Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The axial emergency braking ramp has been activated for the specified axis/spindle The following causes are possible for activation of the emergency braking ramp: Alarm 26052: Path velocity for auxiliary function output too high Alarm 1012 : System error with ID 550006 Alarm 1016 : System error with ID 550003, 550005 and 550010 Context-sensitive braking request with priority 13 is active
Reaction:	NC Start disable in this channel. Local alarm reaction. The NC switches to follow-up mode. Interface signals are set. Alarm display.

Remedy:	Please inform the authorized personnel/service department. Remove or reset the cause of the alarm.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21650 [Channel %1:] Axis %2 overlaid motion not allowed

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	An overlaid motion was requested for the axis, however, this is not allowed due to the MD32074 $MA_FRAME_OR_CORRPOS_NOTALLOWED.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Deselect the overlaid motion or change MD32074 $MA_FRAME_OR_CORRPOS_NOTALLOWED.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21660 [Channel %1:] Block %2 axis %3 conflict between SYNACT: $AA_OFF and CORROF

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name
Definitions:	When deselecting the position offset ($AA_OFF) via the part program command CORROF (<axis>, "AA_OFF") an active synchronized action is detected that immediately sets $AA_OFF for the axis (DO_$AA_OFF [<axis>] =<value>). Deselection is executed and $AA_OFF not set again.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

21665 [Channel %1:] $AA_TOFF cleared

Parameters:	%1 = Channel number
Definitions:	If the tool position is changed with RESET and $AA_TOFF is active during RESET, the position offset ($AA_TOFF) is cleared.
Reaction:	Correction block is reorganized. Local alarm reaction. Interface signals are set. Alarm display. NC Stop on alarm at block end.
Remedy:	Modify the RESET setting in $AA_TOFF_MODE.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

21670 [Channel %1:] Block %2 illegal change of tool direction with $AA_TOFF active

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	If an offset has been activated in tool direction by means of $AA_TOFF[i], no block is allowed to be activated in which the offset axis assignment i is modified (plane change, tool change cutting tool <=> turning tool, transformation change, TRAFOOF, TCARR=0, geometry axis exchange)
Reaction:	Correction block is reorganized.

Local alarm reaction.
Interface signals are set.
Alarm display.
NC Stop on alarm at block end.

Remedy:	- Modify part program - Program TOFFOF()
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

21700 [Channel %1:] Block %3 axis %2 touch probe already deflected, edge polarity not possible

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number
Definitions:	The probe programmed under the keyword MEAS or MEAW is already deflected and has switched. For a further measuring operation, the probe signal must first be canceled (quiescent state of the probe). The axis display is of no significance at the present time but an axis-specific evaluation has been planned for later stages of development.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Verify the starting position of the measuring operation or check the probe signals in the PLC interface DB2700 DBX1.0 / .1 (Probe actuated key 1/key 2). Are the cables and connectors in good order?
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21701 [Channel %1:] Block %3 axis %2 measurement not possible

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number
Definitions:	Measurement level 2 (MEASA, MEAWA, MEAC). There is an error in the programmed measurement task. Possible causes: - Invalid measurement mode - Invalid probe - Invalid encoder - Invalid number of measurement signal edges - Identical measurement signal edges are only programmable in mode 2 - Invalid FIFO number - Mismatch between the number of FIFOs programmed and the number of probes used in the measurement task. Further causes: A measurement task is already active (e.g. from a synchronized action).
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Correct the measurement tasks.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21702 [Channel %1:] Block %3 axis %2 measurement aborted

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

Definitions:	The measurement block has ended (the programmed end position of the axis has been reached) but the activated touch probe has not yet responded. Measurement level 2 (MEAWA, MEASA, MEAC) Measured values cannot be converted to the workpiece coordinate system. The measured values of the GEO axes programmed in the measurement task are only available in the machine coordinate system. Causes: Not all GEO axes were programmed in the measurement task. At least one measured value is therefore missing for conversion back into the workpiece coordinate system. Further causes: The measurement tasks programmed for all GEO axis are not identical.
Reaction:	Alarm display.
Remedy:	Verify the traversing movement in the measurements block. - Is it necessary in all cases for the activated probe to have switched up to the specified axis position? - Are the probe, cable, cable distributor, terminal connections in good order? Either program all GEO axes explicitly or program the traversing movement with the POS[axis] command.
Program Continuation:	Clear alarm with the Delete key or NC START.

21703   [Channel %1:] Block %3 axis %2 touch probe not deflected, illegal edge polarity

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number
Definitions:	The selected probe is not (!) deflected and therefore cannot record any measured value from the deflected to the non-deflected state. Measurement level 2 (MEAWA, MEASA, MEAC) The degree of deflection of the probe at the start of the measurement task is identical to the first programmed measurement signal edge. The test is only performed in mode 2.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	- Check probe - Check start positioning for measuring - Check program
Program Continuation:	Clear alarm with the RESET key. Restart part program

21740   Output value at analog output no. %1 has been limited

Parameters:	%1 = No. of output
Definitions:	The value range of the analog output n is limited by MD10330 $MN_FASTIO_ANA_OUTPUT_WEIGHT[n].
Reaction:	Alarm display.
Remedy:	With $A_OUTA[..] = x no greater values can be programmed than permitted in the respective machine data.
Program Continuation:	Clear alarm with the Delete key or NC START.

21760   [Channel %1:] Block %2 motion synchronous action: %3 too many auxiliary functions programmed

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID
Definitions:	The number of programmed auxiliary functions has exceeded the maximum permissible amount. This alarm can occur in conjunction with motion synchronous actions: The maximum number of auxiliary functions must not be exceeded in motion block and motion synchronous actions.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.

	NC Stop on alarm.
Remedy:	Modify part program.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

21800   [Channel %1:] Workpiece setpoint %2 reached

Parameters:	%1 = Channel number %2 = Workpiece setpoint
Definitions:	This alarm is activated via MD27880 $MC_PART_COUNTER, bit 1: The number of counted workpieces ($AC_ACTUAL_PARTS or $AC_SPECIAL_PARTS) is equal or already greater than the programmed value for the number of required workpieces ($AC_REQUIRED_PARTS). At the same time, the channel VDI signal "Workpiece setpoint reached" is output. The value for the number of counted workpieces ($AC_ACTUAL_PARTS) is reset, while the value of $AC_SPECIAL_PARTS is retained. Note: The setpoint/actual comparisons of the workpieces are only made after an NC start under the condition that $AC_REQUIRED_PARTS > 0. If $AC_REQUIRED_PARTS has a negative value, all workpiece counts activated through MD27880 $MC_PART_COUNTER are frozen at the values they have reached, and the nominal/actual comparison is discontinued.
Reaction:	NC not ready. Interface signals are set. Alarm display.
Remedy:	No program interrupt. Delete alarm display.
Program Continuation:	Clear alarm with the Delete key or NC START.

22000   [Channel %1:] Block %2 Spindle %3 Gear stage change in %4 not possible

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle number %4 = Gear stage
Definitions:	A gear stage change for the spindle will not be possible, if: - thread cutting (G33, G34, G35) is active - the spindle is active as master or slave spindle in a coupling - the spindle is being positioned
Reaction:	Interpreter stop NC Start disable in this channel. NC Stop on alarm. Interface signals are set. Alarm display.
Remedy:	The gear stage is to be set prior to the corresponding machining step. If it is necessary, however, to change the gear stage within one of the above mentioned functions, this function must be switched off for the time of the gear stage change. Thread cutting is deselected with G1; synchronous spindle coupling is switched off with COUPOF; the spindle positioning operation is exited with M3, M4 or M5.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

22005   [Channel %1:] Block %2 motion synchronous action %3 spindle %4 selected gear stage not installed

Parameters:	%1 = Channel number %2 = Block number, line number %3 = Synact ID %4 = Spindle number
Definitions:	The first gear stage data block is active. The required gear stage is not installed in the 1st gear stage data block. The number of installed gear stages is configured in MD35090 $MA_NUM_GEAR_STEPS. Examples for the occurrence of the alarm with 3 gear stages installed (MD35090 $MA_NUM_GEAR_STEPS = 3): * ...DO M44 or DO 45 was programmed in synchronized action for the spindle concerned.

	* ...DO M70 was programmed and MD35014 $MA_GEAR_STEP_USED_IN_AXISMODE was larger than 3.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify part program: Only those valid gear stages can be entered which have also been installed according to MD35090 $MA_NUM_GEAR_STEPS. Limit M70 configuration (MD 35014 $MA_GEAR_STEP_USED_IN_AXISMODE) to MD35090 $MA_NUM_GEAR_STEPS.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

22010   [Channel %1:] Block %3 spindle %2 actual gear stage differs from requested gear stage

Parameters:	%1 = Channel number %2 = Spindle number %3 = Block number, label
Definitions:	The requested gear stage change has been concluded. The actual gear stage reported by the PLC as being engaged is not the same as the required gear stage called for by the NC. Note: Wherever possible, the requested gear stage should always be engaged.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Correct the PLC program.
Program Continuation:	Clear alarm with the Delete key or NC START.

22011   [Channel %1:] Block %3 spindle %2 change to programmed gear stage not possible

Parameters:	%1 = Channel number %2 = Spindle number %3 = Block number, label
Definitions:	With the 'DryRun', 'ProgramTest' and 'SearchRunByProgTest' functions deselected, it is not possible in the Repos module to carry out a gear stage change to a previously programmed gear stage. This is the case, if the spindle is in the deselection block not active in speed control mode, as a slave axis or in a transformation. Execution of a gear stage change is avoided if the above mentioned functions are deselected by resetting bit 2 of MD35035 $MA_SPIND_FUNCTION_MASK.
Reaction:	Alarm display.
Remedy:	Change deselection block or block search target block to speed control mode (M3, M4, M5, SBCOF). Set bit 2 of MD35035 $MA_SPIND_FUNCTION_MASK to 0.
Program Continuation:	Clear alarm with the Delete key or NC START.

22020   [Channel %1:] Block %3 spindle %2 gear step change position not reached

Parameters:	%1 = Channel number %2 = Spindle number %3 = Block number, label
Definitions:	Through the configuration of MD35010 $MA_GEAR_STEP_CHANGE_ENABLE[AXn] = 2, the spindle is traversed to the position stored in MD35012 $MA_GEAR_STEP_CHANGE_POSITION[AXn] before the actual gear step change. The required gear step change position has not been reached.
Reaction:	Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Correct sequence in the PLC.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22022 [Channel %1:] Block %2 spindle %3 gear stage %4 is expected for axis mode.

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle %4 = Gear stage
Definitions:	The gear stage required for axis mode has not been installed. A gear stage has been configured in MD35014 $MA_GEAR_STEP_USED_IN_AXISMODE, in which the spindle is to be in axis mode. This gear stage is checked whenever the spindle is switched into axis mode. The configured gear stage is compared with the gear stage output by the PLC (NC/PLC interface signal DB380x DBX2000.0 - .2 (Actual gear stage A through C)). This alarm will be output if the gear stages are not the same.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Program M70 before the switch to axis mode. The gear stage configured in MD35014 $MA_GEAR_STEP_USED_IN_AXISMODE is then automatically loaded. No gear stage change is required if the configured gear stage is aleady active. M40 remains active beyond the gear stage change. Consider MD20094 $MC_SPIND_RIGID_TAPPING_M_NR.
Program Continuation:	Clear alarm with the Delete key or NC START.

22024 [Channel %1:] Block %2 Spindle %3 tapping: PLC signal 'invert M3/M4' changed after %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Spindle %4 = Value
Definitions:	When loading a G331 block it was detected that the NC/PLC interface signal DB380x DBX2001.6 (invert M3/M4) had changed during part program execution. An alarm was output to prevent a tool break. The current value of the NC/PLC interface signal is displayed as parameter 4.
Reaction:	Channel not ready. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	A change in the NC/PLC interface signal DB380x DBX2001.6 (invert M3/M4) during part program execution should be avoided. If MD35035 SPIND_FUNCTION_MASK bit 22 is set, the NC/PLC interface signal DB380x DBX2001.6 (invert M3/M4) is then no longer evaluated during tapping with G331, G332. The alarm is no longer output. Notice! Setting bit 22 means a change in function.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22040 [Channel %1:] Block %3 spindle %2 is not referenced with zero marker

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	The current position is not referenced with the measuring system position although reference is made to it.
Reaction:	Alarm display.
Remedy:	Correct NC part program. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on. If this has been intentionally programmed, the alarm can be suppressed in the cyclic check with position control already enabled by means of MD11410 $MN_SUPPRESS_ALARM_MASK Bit21 = 1.
Program Continuation:	Clear alarm with the Delete key or NC START.

22050	**[Channel %1:] Block %3 spindle %2 no transition from speed control mode to position control mode**
Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	- An oriented spindle stop (SPOS/SPOSA) has been programmed or the position control of the spindle was switched on with SPCON but no spindle encoder has been defined. - When switching on the position control, the spindle speed is greater than the limiting speed of the measuring system.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Spindle without attached encoder: Any NC language elements requiring the encoder signals must not be used. Spindle with attached encoder: Enter the number of spindle encoders used in the MD30200 $MA_NUM_ENCS.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22051	**[Channel %1:] Block %3 spindle %2 reference mark not found**
Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	When referencing, the spindle turned through a greater distance than given in the axis-specific MD34060 $MA_REFP_MAX_MARKER_DIST, without receiving a reference mark signal. The check is performed for spindle positioning with SPOS or SPOSA when the spindle has not previously run with speed control (S=...).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check and correct the MD34060 $MA_REFP_MAX_MARKER_DIST. The value entered states the distance in [mm] or [degrees] between 2 zero markers.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22052	**[Channel %1:] Block %3 spindle %2 no standstill on block change**
Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	The displayed spindle has been programmed as spindle or as axis even though a positioning operation is still running from the previous block (with SPOSA ... spindle positioning beyond block limits). Example: N100 SPOSA [2] = 100 : N125 S2 = 1000 M2 = 04 ; Error, if spindle S2 from block N100 is still running!
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Before programming the spindle/axis again using the SPOSA instruction, a WAITS command should be activated in order to wait for the programmed spindle position. Example: N100 SPOSA [2] = 100 : N125 WAITS (2) N126 S2 = 1000 M2 = 04
Program Continuation:	Clear alarm with the RESET key. Restart part program

22053 [Channel %1:] Block %3 spindle %2 reference mode not supported

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	In the case of SPOS/SPOSA with an absolute encoder, only the referencing mode MD34200 $MA_ENC_REFP_MODE = 2 is supported! SPOS/SPOSA does not support MD34200 $MA_ENC_REFP_MODE = 6 at all!
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Modify setting of MD34200 $MA_ENC_REFP_MODE or change to JOG+REF and then reference.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22055 [Channel %1:] Block %3 spindle %2 configured positioning speed is too high

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	The current position is not referenced with the measuring system position although reference is made to it.
Reaction:	Alarm display.
Remedy:	Correct NC part program. Create the zero mark synchronization by positioning, by rotation (at least 1 revolution) in speed control mode or G74 before switching the alarm generating function on.
Program Continuation:	Clear alarm with the Delete key or NC START.

22060 [Channel %1:] Position control expected for axis/spindle %2

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The programmed coupling type (DV, AV) or the programmed function requires position control.
Reaction:	Alarm display.
Remedy:	Activate position control, e.g. by programming SPCON.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

22062 [Channel %1:] Axis %2 reference point approach: zero marker search velocity (MD) is not reached

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The configured zero marker search velocity is not reached.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check active spindle speed limitations. Configure a lower zero marker search velocity MD34040 $MA_REFP_VELO_SEARCH_MARKER. Check the tolerance range for the actual velocity MD35150 $MA_SPIND_DES_VELO_TOL. Set a different referencing mode MD34200 $MA_ENC_REFP_MODE != 7.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22064 [Channel %1:] Axis %2 reference point approach: zero marker search velocity (MD) is too high

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The configured zero marker search velocity is too high. The encoder limit frequency is exceeded for the active measuring system.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Configure a lower zero marker search velocity MD34040 $MA_REFP_VELO_SEARCH_MARKER. Check the encoder frequency configuration MD36300 $MA_ENC_FREQ_LIMIT and MD36302 $MA_ENC_FREQ_LIMIT_LOW. Set a different referencing mode MD34200 $MA_ENC_REFP_MODE=7.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22100 [Channel %1:] Block %3 spindle %2 chuck speed exceeded

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Block number, label
Definitions:	The actual spindle speed is higher than the maximum speed configured in MD35100 $MA_SPIND_VELO_LIMIT plus a tolerance of 10 percent (fixed setting). The alarm should not occur after correct optimization of the drive actuator and gear configuration. This alarm can be reconfigured with MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready to operate) to 'BAG not ready'. Note: Reconfiguring affects all alarms with alarm response 'Chan not ready'.
Reaction:	Mode group not ready. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Check the setup and optimization data of the drive actuator in accordance with the Installation and Start-up Guide and make corrections. Increase the tolerance window in MD35150 $MA_SPIND_DES_VELO_TOL.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

22200 [Channel %1:] Spindle %2 axis stopped during tapping

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	When tapping with compensating chuck (G63) the drilling axis was stopped via the NC/PLC interface and the spindle continues to rotate. The thread and possibly also the tap were damaged as a result.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Provide an interlock in the PLC user program so that no axis stop can be initiated when tapping is active. If the tapping operation is to be terminated under critical machine conditions, the spindle and the axis should be stopped simultaneously if at all possible. Slight differences are then accommodated by the compensating chuck.
Program Continuation:	Clear alarm with the RESET key. Restart part program

22250 [Channel %1:] Spindle %2 axis stopped during thread cutting

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The thread cutting axis has been stopped while a thread block was active. The stop can be caused by VDI signals that cause the feed to be interrupted.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check the axis-specific/spindle-specific stop DB380x DBX4.3 (Spindle stop).
Program Continuation:	Clear alarm with the RESET key. Restart part program

22260 [Channel %1:] Spindle %2 thread might be damaged

Parameters:	%1 = Channel number %2 = Axis name %3 = Block number
Definitions:	When DECODING SINGLE BLOCK has been selected and there is a chain of thread blocks, then machining pauses occur at the block limits until the next block is executed with the new NC Start. In normal single block mode, the program is stopped by a higher-level logic only at the block boundaries at which no contour distortions or contour errors can occur. With chained thread blocks, this is the last thread block!
Reaction:	Alarm display.
Remedy:	If only one thread block has been programmed, the alarm message can be ignored. If there are several consecutive thread blocks, this machining section must not be executed in the automatic DECODING SINGLE BLOCK mode.
Program Continuation:	Clear alarm with NC START or RESET key and continue the program.

22270 [Channel %1:] Block %2 thread cutting: Maximum speed axis %3 exceeded %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name %4 = Velocity
Definitions:	Thread cutting with G33, G34, G35: The thread axis (pitch axis) velocity calculated exceeds the maximum permissible axis velocity MD32000 $MA_MAX_AX_VELO. The calculated axis velocity is displayed. The velocity of the thread axis is dependent upon: - The current spindle speed - The programmed thread pitch - The programmed thread pitch change and thread length (G34, G35) - The spindle override (path and individual axis overrides are ineffective)
Reaction:	Alarm display.
Remedy:	Reduce the spindle speed or thread pitch (thread pitch change).
Program Continuation:	Clear alarm with the Delete key or NC START.

22271 [Channel %1:] Block %2 thread cutting: Maximum speed axis %3 exceeded %4

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name %4 = Velocity
Definitions:	Thread cutting with G33, G34, G35: The thread axis (pitch axis) velocity calculated exceeds the maximum permissible axis velocity MD32000 $MA_MAX_AX_VELO. The calculated axis velocity is displayed. The velocity of the thread axis is dependent upon: - The current spindle speed - The programmed thread pitch - The programmed thread pitch change and thread length (G34, G35)

- The spindle override (path and individual axis overrides are ineffective)

Reaction:	Alarm display.
Remedy:	Reduce the spindle speed or thread pitch (thread pitch change).
Program Continuation:	Clear alarm with the Delete key or NC START.

22272 [channel %1:] block %2 axis %3 thread cutting: block length %4 too short for predefined thread pitch

Parameters:	%1 = Channel number %2 = Block number, label %3 = Axis name %4 = Block length
Definitions:	Thread cutting with G33, G34, G35: Block length too short for predefined thread pitch. Block length is displayed.
Reaction:	Alarm display.
Remedy:	Reduce thread pitch (thread pitch change).
Program Continuation:	Clear alarm with the Delete key or NC START.

22275 [Channel %1:] Block %2 zero velocity of thread axis at position %3 reached

Parameters:	%1 = Channel number %2 = Block number, label %3 = Position
Definitions:	An axis standstill was reached at the specified position during thread cutting with G35 due to the linear decrease in the thread pitch. The standstill position of the thread axis depends on: - Programmed thread pitch decrease - Thread length
Reaction:	Alarm display.
Remedy:	Change at least one of the above factors.
Program Continuation:	Clear alarm with the Delete key or NC START.

22280 [Channel %1:] In block %2: Prog. acceleration path too short %3, %4 required

Parameters:	%1 = Channel number %2 = Block number, label %3 = Prog. acceleration path %4 = Required acceleration path
Definitions:	In order to stay within the programmed acceleration path, the acceleration caused an overload on the thread axis. In order to accelerate the axis with the programmed dynamic response, the length of the acceleration path must be at least as large as the value in parameter %4. The alarm is of the technological type and is output whenever bit 2 in MD11411 $MN_ENABLE_ALARM_MASK is enabled. The HMI softkey 'Technology support' sets and clears this bit in the MD.
Reaction:	Alarm display.
Remedy:	Modify part program or reset MD11411 $MN_ENABLE_ALARM_MASK bit 2.
Program Continuation:	Clear alarm with the Delete key or NC START.

22290 [Channel %1:] Spindle operation for transformed spindle/axis %2 not possible (reason: error code %3).

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Error code
Definitions:	It is impermissible to start a spindle as long as it is being used by a transformation. Reason: spindle usage in a transformation requires axis operation, which must not be exited. This alarm may have the following reasons: - Error code 1 : M3, M4 or M5 per synchronized action;

- Error code 2 : M41 through M45 per synchronized action;
- Error code 3 : SPOS, M19 per synchronized action;
- Error code 11 : DB380x DBX5006.0 (Spindle stop);
- Error code 12 : DB380x DBX5006.1 (Spindle start clockwise rotation);
- Error code 13 : DB380x DBX5006.2 (Spindle start counterclockwise rotation;
- Error code 14 : DB380x DBX5006.4 (Spindle positioning).

Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Resolve the conflict, for example by deactivating transformation prior to spindle start.
Program Continuation:	Clear alarm with the Delete key or NC START.

22295   [Channel %1:] Spindle %2 DBB30 function not possible (cause: error code %3)

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Error code
Definitions:	The function request by PLC via DBB30 interface could not be executed. The cause is specified by the error code. Error codes: - Error code 1 : internal use - Error code 2 : internal use - Error code 3 : internal use - Error code 4 : internal use - Error code 5 : Switchover to command axis not possible - Error code 6 : Switchover to PLC axis not possible - Error code 20 : internal use - Error code 21 : internal use - Error code 22 : internal use - Error code 23 : MD 30132 IS_VIRTUAL_AX has been set - Error code 50 : internal use - Error code 51 : internal use - Error code 70 : internal use
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Remedy conflict.
Program Continuation:	Clear alarm with the Delete key or NC START.

22296   [Channel %1:] Spindle %2 Error on gear stage change (cause: error code %3)

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Error code
Definitions:	An error occurred during gear stage change. The cause is specified by the error code. Error codes: - Error code 1 : internal use - Error code 2 : internal use - Error code 3 : internal use - Error code 4 : internal use - Error code 5 : Switchover to command axis not possible - Error code 6 : Switchover to PLC axis not possible - Error code 20 : internal use - Error code 21 : internal use - Error code 22 : internal use - Error code 23 : MD 30132 IS_VIRTUAL_AX has been set - Error code 50 : internal use - Error code 51 : internal use - Error code 70 : internal use
Reaction:	NC Start disable in this channel.

Interface signals are set.
Alarm display.

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

22297 [Channel %1:] Spindle %2 FC18 function not possible (cause: error code %3)

Parameters:	%1 = Channel number %2 = Axis name, spindle number %3 = Error code
Definitions:	The function request by PLC via FC18 interface could not be executed. The cause is specified by the error code. Error codes: - Error code 1 : internal use - Error code 2 : internal use - Error code 3 : internal use - Error code 4 : internal use - Error code 5 : Switchover to command axis not possible - Error code 6 : Switchover to PLC axis not possible - Error code 20 : internal use - Error code 21 : internal use - Error code 22 : internal use - Error code 23 : MD 30132 IS_VIRTUAL_AX has been set - Error code 50 : internal use - Error code 51 : internal use - Error code 70 : internal use
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Remedy conflict.
Program Continuation:	Clear alarm with the Delete key or NC START.

22321 [Channel %1:] Axis %2 PRESET not allowed during traverse motion

Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	A preset command was sent from the HMI or PLC while an axis was traveling in JOG mode.
Reaction:	Interface signals are set. Alarm display.
Remedy:	Wait until the axis is stationary.
Program Continuation:	Clear alarm with the Delete key or NC START.

22322 [Channel %1:] Axis %2 PRESET: illegal value

Parameters:	%1 = Channel number %2 = Axis name, spindle number
Definitions:	The entered Preset value is too large (number format overflow).
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Use more realistic (smaller) Preset values.
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

22400 [Channel %1:] Option 'contour handwheel' not set

Parameters:	%1 = Channel number

Definitions:	The function 'contour handwheel' was activated without the neccesary option.
	If the alarm occurs
	- on selection of the contour handwheel via the PLC, then the contour handwheel has to be deselected in order to continue with the program
	- on account of programming FD=0, then the program can be corrected and continued with the compensation block and NCSTART.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department.
	- Set option
	- Cancel the activation of the function 'contour handwheel'
	- Modify part program.
Program Continuation:	Clear alarm with the Delete key or NC START.

25000 Axis %1 hardware fault of active encoder

Parameters:	%1 = Axis name, spindle number
Definitions:	The signals of the currently active position actual value encoder (NC/PLC interface signal DB380x DBX1.5 = 1 (Position measuring system 1) or DB380x DBX1.6 = 1 (Position measuring system 2)) are missing, do not have the same phase, or exhibit grounding/short-circuit.
	The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready.
	The NC switches to follow-up mode.
	Channel not ready.
	NC Start disable in this channel.
	Axes of this channel must be re-referenced.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
	Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found.
	Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).
Program Continuation:	Switch control OFF - ON.

25001 Axis %1 hardware fault of passive encoder

Parameters:	%1 = Axis name, spindle number
Definitions:	The signals from the currently inactive position actual value encoder are missing, or they are not of the same phase, or they exhibit grounding/short-circuit.
	MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing Reset alarm by the Cancel alarm 25011.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found. Switch off monitoring with the corresponding interface signal DB380x DBX1.5 / 1.6 = 0 (Position measuring system 1/2).
	Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).
Program Continuation:	Clear alarm with the RESET key. Restart part program

---

25010 Axis %1 pollution of measuring system

Parameters:	%1 = Axis name, spindle number
Definitions:	The encoder used for position control sends a contamination signal (only in measuring systems with contamination signal).
	The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready.
	The NC switches to follow-up mode.

Channel not ready.
NC Start disable in this channel.
Axes of this channel must be re-referenced.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Channel not ready.

Remedy:	Please inform the authorized personnel/service department. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.
	Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25011   Axis %1 pollution of passive encoder

Parameters:	%1 = Axis name, spindle number
Definitions:	The encoder not used for position control sends a contamination signal (only in measuring systems with contamination signal).
	MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Cancel alarm instead of the Reset alarm 25001.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.
	Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).
Program Continuation:	Clear alarm with the Delete key or NC START.

25020   Axis %1 zero mark monitoring of active encoder

Parameters:	%1 = Axis name, spindle number
Definitions:	The alarm can be reprogrammed in MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready.
	The NC switches to follow-up mode.
	Channel not ready.
	NC Start disable in this channel.
	Axes of this channel must be re-referenced.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
	Channel not ready.
Remedy:	Please inform the authorized personnel/service department.
	The differences can result from transmission errors, disturbances, encoder hardware faults or from the evaluation electronics in the encoder used for position control. The actual value branch must therefore be checked:
	1. Transmission path: Check the actual-value connectors for correct contacting, encoder cable for continuity, and also check for short-circuits and grounding (loose contact?).
	2. Encoder pulses: Is the encoder power supply within the tolerance limits?
	3. Evaluation electronics: Replace or reconfigure the drive or encoder module used.
	4. Check MD34220 $MA_ENC_ABS_TURNS_MODULO and Sinamics drive parameter P0979 subindex 5 (or 15,25). They have to be the same for correct handling of the encoder data.
	Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING [n] to 0 or 100 (n = encoder number: 1, 2).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25021   Axis %1 zero mark monitoring of passive encoder

Parameters:	%1 = Axis name, spindle number

Definitions:	Monitoring relates to the encoder that is not used by the position control. (NC-PLC interface signal DB380x DBX1.5 = 0 (Position measuring system 1) or DB380x DBX1.6 = 0 (Position measuring system 2)) More detailed explanations are similar to those for alarm 25020.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. The differences can result from transmission errors, disturbances, encoder hardware faults or from the evaluation electronics in the encoder not used for position control. The actual value branch must therefore be checked: 1. Transmission path: Check the actual-value connectors for correct contacting, encoder cable for continuity, and also check for short-circuits and grounding (loose contact?). 2. Encoder pulses: Is the encoder power supply within the tolerance limits? 3. Evaluation electronics: Replace or reconfigure the drive or encoder module used. 4. Check MD34220 $MA_ENC_ABS_TURNS_MODULO and Sinamics drive parameter P0979 subindex 5 (or 15,25). Both have to be the same for correct handling of the encoder data. Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 0 or 100 (n = encoder number: 1, 2).
Program Continuation:	Clear alarm with the Delete key or NC START.

25022   Axis %1 encoder %2 warning %3

Parameters:	%1 = Axis name, spindle number %2 = Encoder number %3 = Error fine coding
Definitions:	This alarm only occurs with absolute encoders: a. Warning notice of missing absolute encoder adjustment (on the SIMODRIVE 611D or with PROFIdrive drives), that is if MD34210 $MA_ENC_REFP_STATE equals 0. In this case, fine error code 0 is returned. b. If, on the SIMODRIVE 611D only, zero mark monitoring has been activated for the absolute encoder (see MD36310 $MA_ENC_ZERO_MONITORING): In this case, the absolute position of the absolute encoder could not be read without error: Breakdown of fine error codes: (Bit 0 not used) Bit 1 Parity error Bit 2 Alarm bit of the encoder Bit 3 CRC error Bit 4 Timeout - start bit for EnDat transfer is missing This alarm is only displayed, as the absolute position itself is not required at this time for control/contour. A frequent occurrance of this alarm indicates that the absolute encoder transfer or the absolute encoder itself is faulty, and that an incorrect absolute value could be determined in one of the next encoder selection or power on situations.
Reaction:	Alarm display.
Remedy:	a. Verify encoder adjustment (machine reference) or readjust encoder. b. Replace the encoder, replace or screen the encoder cable (or deactivate zero mark monitoring).
Program Continuation:	Clear alarm with the Delete key or NC START.

25030   Axis %1 actual velocity alarm limit

Parameters:	%1 = Axis name, spindle number
Definitions:	If the axis has at least one active encoder, then the actual speed of the axis is cyclically checked in the IPO cycle. If there are no errors, the actual velocity can never become greater than specified in the axis-specific MD36200 $MA_AX_VELO_LIMIT (threshold for velocity monitoring). This threshold value in [mm/min, rev/min] is input by an amount that is about 5 to 10% greater than that which can occur at maximum traversing velocity. Drive errors can result in the velocity being exceeded and the alarm is then triggered. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.

Remedy:	Please inform the authorized personnel/service department. - Check the speed setpoint cable (bus cable). - Check the actual values and direction of position control. - Change the position control direction if the axis rotates uncontrollably -> axis-specific MD32110 $MA_ENC_FEEDBACK_POL [n] = < -1, 0, 1 >. - Increase the monitoring limit value in MD 36200 $MA_AX_VELO_LIMIT.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25031   Axis %1 actual velocity warning limit

Parameters:	%1 = Axis name, spindle number
Definitions:	The present velocity actual value is more than 80% of the limit value defined in the machine data. (Internal test criterion activated by MD36690 $MA_AXIS_DIAGNOSIS, bit0)
Reaction:	Alarm display.
Remedy:	-
Program Continuation:	Clear alarm with the Delete key or NC START.

25040   Axis %1 standstill monitoring

Parameters:	%1 = Axis name, spindle number
Definitions:	The NC monitors to ensure that the position is held at zero speed. Monitoring is started after a time that can be set for a specific axis in the MD36040 $MA_STANDSTILL_DELAY_TIME after interpolation has ended. A constant check is made to determine whether the axis remains within the tolerance range given in MD36030 $MA_STANDSTILL_POS_TOL. The following cases are possible: 1. The NC/PLC interface signal DB380x DBX2.1 (Servo enable) is zero because the axis has jammed mechanically. Due to mechanical influences (e.g. high machining pressure), the axis is pushed outside the permissible position tolerance. 2. With closed position control loop (without jamming) - NC/PLC interface signal DB380x DBX2.1 (Servo enable) is "1" - the axis is pushed away from its position by mechanical forces with a small gain in the position control loop. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. - Check MD36040 $MA_STANDSTILL_DELAY_TIME and MD36030 $MA_STANDSTILL_POS_TOL; increase if necessary. The value must be greater than the machine data "Exact stop - coarse" (MD36000 $MA_STOP_LIMIT_COARSE). - Estimate machining forces and reduce if necessary by setting a lower feed or a higher rotational speed. - Increase clamping pressure. - Increase the gain in the position control loop by improved optimization (Kv factor MD32200 $MA_POSCTRL_GAIN, SIMODRIVE611D drive).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25050   Axis %1 contour monitoring

Parameters:	%1 = Axis name, spindle number
Definitions:	The NCK calculates for each interpolation point (setpoint) of an axis the actual value that should result based on an internal model. If this calculated actual value and the true machine actual value differ by a larger amount than given in the MD36400 $MA_CONTOUR_TOL, then the program is aborted and the alarm message is issued. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction:	Mode group not ready.
	The NC switches to follow-up mode.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
	Channel not ready.
Remedy:	Please inform the authorized personnel/service department.
	- Check whether the tolerance value set in MD36400 $MA_CONTOUR_TOL is too small.
	- Check optimization of the position controller (Kv factor in the MD32200 $MA_POSCTRL_GAIN) to establish whether the axis follows the given setpoint without overshooting. Otherwise, the speed controller optimization must be improved or the Kv servo gain factor must be reduced.
	- Improvement of speed controller optimization
	- Check the mechanics (smooth running, inertial masses).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25060 Axis %1 speed setpoint limitation

Parameters:	%1 = Axis name, spindle number
Definitions:	The speed setpoint has exceeded its upper limit for a longer period than allowed.
	The maximum speed setpoint is limited to a certain percentage by the axis-specific MD36210 $MA_CTRLOUT_LIMIT. The input value of 100% corresponds to the rated speed of the motor and hence the rapid traverse velocity (exemplary default value: 840D=110%).
	If the values are exceeded for a short time, then this is tolerated provided they do not last longer than allowed for in the axis-specific MD36220 $MA_CTRLOUT_LIMIT_TIME. The setpoint is limited during this time to the maximum value that has been set in (MD36210 $MA_CTRLOUT_LIMIT).
	The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready.
	The NC switches to follow-up mode.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
	Channel not ready.
Remedy:	Please inform the authorized personnel/service department. This alarm should not occur if the drive controller has been set correctly and the machining conditions are those that normally prevail.
	- Check actual values: Local sluggishness of the carriage, speed dip by torque surge due to contact with workpiece/tool, travel against fixed obstacle, etc.
	- Check direction of position control: Does the axis continue to rotate without control (not on SIMODRIVE 611D drives)?
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25070 Axis %1 drift value too large

Parameters:	%1 = Axis name, spindle number
Definitions:	Only with analog drives!
	The permissible maximum value of drift (internal, integrated drift value of automatic drift compensation) was exceeded during the last compensation operation. The permissible maximum value is defined in the axis-specific MD36710 $MA_DRIFT_LIMIT. The drift value itself is not limited.
	Automatic drift compensation: MD36700 $MA_DRIFT_ENABLE=1
	The difference between actual and setpoint position (drift) is checked cyclically in the IPO cycle when the axes are at zero speed. The difference is automatically compensated to zero by slowly integrating an internal drift value.
	Drift compensation by hand: MD36700 $MA_DRIFT_ENABLE=0
	A static offset can be added to the speed setpoint in the MD36720 $MA_DRIFT_VALUE. This is not included in the drift monitoring because it acts like a voltage work offset.
Reaction:	Alarm display.

Remedy:	Please inform the authorized personnel/service department. Adjust the drift compensation with the automatic drift compensation switched off at the drive until the position lag is approximately zero. Then reactivate the automatic drift compensation in order to balance out the dynamic drift changes (effects of heating up).
Program Continuation:	Clear alarm with the Delete key or NC START.

25080   Axis %1 positioning monitoring

Parameters:	%1 = Axis name, spindle number
Definitions:	For blocks in which "exact stop" is effective, the axis must have reached the exact stop window after the positioning time given in the axis-specific MD36020 $MA_POSITIONING_TIME. Exact stop coarse: MD36000 $MA_STOP_LIMIT_COARSE Exact stop fine: MD36010 $MA_STOP_LIMIT_FINE The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Check whether the exact stop limits (coarse and fine) correspond to the dynamic possibilities of the axis, otherwise increase them, if necessary in connection with the positioning time set in MD36020 $MA_POSITIONING_TIME. Check speed controller/position controller optimization; select highest possible gain. Check setting of Kv factor (MD32200 $MA_POSCTRL_GAIN) and increase, if required.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25100   Axis %1 measuring system switchover not possible

Parameters:	%1 = Axis name, spindle number
Definitions:	The prerequisites are not satisfied for the required encoder switchover: 1. The newly selected encoder must be in the active state: (DB380x DBX1.5 / 1.6 = 1 (Position measuring system 1/2). 2. The actual value difference between the two encoders is greater than the value in the axis-specific MD36500 $MA_ENC_CHANGE_TOL ("Maximum tolerance for position actual value switchover"). Activation of the measuring system concerned takes place in accordance with the NC/PLC interface signals DB380x DBX1.5 (Position measuring system 1) and DB380x DBX1.6 (Position measuring system 2), i.e. the position control is now operated with this measuring system. The other measuring system is switched over to follow-up mode. If both interface signals are set to "1", then only the 1st measuring system is active; if both interface signals are set to "0", the axis is parked. Changeover takes place as soon as the interface signals have changed, even if the axis is in motion.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. When referencing the active position actual value encoder, the actual value system of the inactive encoder is set to the same reference point value as soon as phase 3 has been concluded. A later positional difference between the two actual value systems can have occurred only as the result of an encoder defect or a mechanical displacement between the encoders. - Check the encoder signals, actual value cable, connectors. - Check the mechanical fastenings (displacement of the measuring head, mechanical twisting possible). - Increase the axis-specific MD 36500 $MA_ENC_CHANGE_TOL. Program continuation is not possible. The program must be aborted with "RESET", then program execution can be reinitiated with NC START, if necessary at the interruption point after "Block search with/without calculation".
Program Continuation:	Clear alarm with the RESET key. Restart part program

25105   Axis %1 measuring systems differ considerably

Parameters:	%1 = Axis name, spindle number
Definitions:	The two measuring systems differ considerably, i.e. the cyclically monitored actual value difference between the two measuring systems is greater than the associated tolerance value set in the machine data MD36510 $MA_ENC_DIFF_TOL. This can only occur when both measuring systems are active (MD30200 $MA_NUM_ENCS = 2) and referenced. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Check machine data for the active, selected encoders. Check the machine data relating to encoder (MD36510 $MA_ENC_DIFF_TOL) tolerance.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25110   Axis %1 selected encoder not available

Parameters:	%1 = Axis name, spindle number
Definitions:	The selected encoder does not correspond to the maximum number of encoders in the axis-specific MD30200 $MA_NUM_ENCS, i.e. the 2nd encoder does not exist.
Reaction:	Alarm display.
Remedy:	Please inform the authorized personnel/service department. Enter the number of actual value encoders used for this axis in the MD30200 $MA_NUM_ENCS ("Number of encoders"). Input value 0: Axis without encoder -> e.g. spindle Input value 1: Axis with encoder -> default setting Input value 2: Axis with 2 encoders -> e.g. direct and indirect measuring system
Program Continuation:	Clear alarm with the Delete key or NC START.

25200   Axis %1 requested set of parameters invalid

Parameters:	%1 = Axis name, spindle number
Definitions:	A new parameter set has been requested for the positioning control. The number of this parameter set is beyond the permissible limit.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Check the axis-specific/spindle-specific interface signals DB380x DBX9.0 - .2 (Select parameter set servo A, B, C). One parameter set includes the following machine data: - MD31050 $MA_DRIVE_AX_RATIO_DENOM [n] - MD31060 $MA_DRIVE_AX_RATIO_NUMERA [n] - MD32200 $MA_POSCTRL_GAIN [n] - MD32800 $MA_EQUIV_CURRCTRL_TIME [n] - MD32810 $MA_EQUIV_SPEEDCTRL_TIME [n] - MD32910 $MA_DYN_MATCH_TIME [n] - MD36200 $MA_AX_VELO_LIMIT [n]
Program Continuation:	Clear alarm with the RESET key. Restart part program

25201   Axis %1 drive fault

Parameters:	%1 = Axis name, spindle number

Definitions:	The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Evaluation of the drive alarms listed above.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

25220   Axis %1 gear ratio changed with ESR enabled

Parameters:	%1 = Axis name, spindle number
Definitions:	Because a change in the gear ratio has an effect on the retraction path traveled by the drive, no changes may be made to the gear ratio during the following periods of time: - Between the last programming of the retraction path with ESRR and the ESR enable - As from ESR enable The following machine data define the gear ratios of an axis MD31050 $MA_DRIVE_AX_RATIO_DENOM MD31060 $MA_DRIVE_AX_RATIO_NUMERA MD31064 $MA_DRIVE_AX_RATIO2_DENOM MD31066 $MA_DRIVE_AX_RATIO2_NUMERA The gear ratio must not be changed during the periods of time described above, e.g. by changing the parameter set.
Reaction:	NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	If ESR enable is missing: - Change the gear ratio before programming the retraction path with ESRR or - Reprogram the retraction path with ESRR after changing the gear ratio Then re-enable ESR.
Program Continuation:	Clear alarm with the RESET key. Restart part program

26000   Axis %1 clamping monitoring

Parameters:	%1 = Axis name, spindle number
Definitions:	The clamped axis has been pushed out of its setpoint position. The permissible difference is defined in the axis-specific MD36050 $MA_CLAMP_POS_TOL. Clamping an axis is activated with the axis-specific interface signal DB380x DBX2.3 (Clamping process active). The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Determine the position deviation to the setpoint position and, depending on the results, either increase the permissible tolerance in the MD or mechanically improve the clamping (e.g. increase clamping pressure).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26001   Axis %1 parameterization error: friction compensation

Parameters:	%1 = Axis name, spindle number

Definitions:	The parameterization of the adaptation characteristic in the quadrant error compensation is not allowed because acceleration value 2 (MD32560 $MA_FRICT_COMP_ACCEL2 is not between acceleration value 1 (MD32550 $MA_FRICT_COMP_ACCEL1) and acceleration value 3 (MD32570 $MA_FRICT_COMP_ACCEL3). The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Check the setting parameters of the quadrant error compensation (friction compensation), if necessary switch off the compensation with MD32500 $MA_FRICT_COMP_ENABLE.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26002   Axis %1 encoder %2 parameterization error: number of encoder marks

Parameters:	%1 = Axis name, spindle number %2 = Encoder number
Definitions:	1. Rotary measuring system (MD31000 $MA_ENC_IS_LINEAR[]==FALSE) The number of encoder marks set in MD31020 $MA_ENC_RESOL[] does not correspond to the value in the drive machine data (SIMODRIVE 611D: MD1005 $MD_ENC_RESOL_MOTOR; PROFIdrive: p979) or zero has been entered in one of the two machine data. 2. Absolute measuring system with EnDat interface (MD30240 $MA_ENC_TYPE[]==4) On absolute encoders, the resolution of the incremental and absolute tracks supplied by the drive is also checked for consistency.
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Adjust machine data. For absolute encoders, possibly pending drive alarms indicating encoder problems should be evaluated. They could be the cause of incorrect entries of MD1022 $MD_ENC_ABS_RESOL_MOTOR/MD1032 $MD_ENC_ABS_RESOL_DIRECT, which the drive reads out of the encoder itself.
Program Continuation:	Switch control OFF - ON.

26003   Axis %1 parameterization error: lead screw pitch

Parameters:	%1 = Axis name, spindle number
Definitions:	The pitch of the ballscrew/trapezoidal leadscrew set in the axis-specific MD31030 $MA_LEADSCREW_PITCH is zero. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Determine the leadscrew pitch (specify the machine manufacturer or pitch measurement with spindle cover removed) and enter it in the MD31030: $MA_LEADSCREW_PITCH (mostly 10 or 5 mm/rev.).

Program Continuation:	Switch control OFF - ON.

26004   Axis %1 encoder %2 parameterization error: grid point distance with linear encoders

Parameters:	%1 = Axis name, spindle number %2 = Encoder number
Definitions:	The scale division of the linear scale set in the axis-specific MD31010 $MA_ENC_GRID_POINT_DIST is zero or differs from the corresponding drive parameters. For a better understanding of the interrelations see the explanations for alarm 26002, which refer to rotary encoders. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Enter the encoder grid point distance according to the data given by the machine (or measuring device) manufacturer in the MD31010 $MA_ENC_GRID_POINT_DIST.
Program Continuation:	Switch control OFF - ON.

26005   Axis %1 parameterization error: output rating

Parameters:	%1 = Axis name, spindle number
Definitions:	For analog drives: The output evaluation of the analog speed setpoint set in the MD32250 $MA_RATED_OUTVAL or in MD 32260 $MA_RATED_VELO is zero. The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. The nominal output voltage in [%] of the maximum setpoint value (10 V) is entered in the MD32250 $MA_RATED_OUTVAL, at which the rated motor speed in [degrees/s] is to be reached (MD32260 $MA_RATED_VELO).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26006   Axis %1 encoder %2 encoder type/output type %3 not possible

Parameters:	%1 = Axis name, spindle number %2 = Encoder number %3 = Encoder type/output type
Definitions:	Not every encoder type or output type can be used with every control or drive variant. Permissible settings: MD30240 $MA_ENC_TYPE = 0 Simulation = 1 Raw signal incremental encoder (SIMODRIVE 611D and PROFIdrive) = 4 Absolute encoder (EnDat with SIMODRIVE 611D; all drive-side absolute encoders supported by PROFIdrive) MD30130 $MA_CTRLOUT_TYPE = 0 Simulation = 1 Standard (SIMODRIVE 611D and PROFIdrive drives) The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction:	Mode group not ready.
	The NC switches to follow-up mode.
	Channel not ready.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
	Channel not ready.
Remedy:	Please inform the authorized personnel/service department.
	Check machine data MD30240 $MA_ENC_TYPE and/or MD30130 $MA_CTRLOUT_TYPE and make the necessary corrections.
Program Continuation:	Switch control OFF - ON.

26014   Axis %1 machine data %2 invalid value

Parameters:	%1 = Axis name, spindle number
	%2 = String: MD identifier
Definitions:	Machine data includes a value that is not valid.
Reaction:	NC not ready.
	The NC switches to follow-up mode.
	Mode group not ready, also effective for single axes.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Repeat entry with correct value and then Power On.
Program Continuation:	Switch control OFF - ON.

26015   Axis %1 machine data %2[%3] invalid value

Parameters:	%1 = Axis name, spindle number
	%2 = String: MD identifier
	%3 = Index: MD array index
Definitions:	Machine data includes a value that is not valid.
Reaction:	NC not ready.
	The NC switches to follow-up mode.
	Mode group not ready, also effective for single axes.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Repeat entry with correct value and then Power On.
Program Continuation:	Switch control OFF - ON.

26016   Axis %1 machine data %2 invalid value

Parameters:	%1 = Axis name, spindle number
	%2 = String: MD identifier
Definitions:	Machine data includes a value that is not valid.
Reaction:	NC not ready.
	The NC switches to follow-up mode.
	Mode group not ready, also effective for single axes.
	NC Start disable in this channel.
	Interface signals are set.
	Alarm display.
	NC Stop on alarm.
Remedy:	Repeat entry with correct value and then Reset.

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

26017   Axis %1 machine data %2[%3] invalid value

Parameters:	%1 = Axis name, spindle number %2 = String: MD identifier %3 = Index: MD array
Definitions:	Machine data includes a value that is not valid.
Reaction:	NC not ready. The NC switches to follow-up mode. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Repeat entry with correct value and then Reset.
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26018   Axis %1 setpoint output drive %2 used more than once

Parameters:	%1 = Axis name, spindle number %2 = Drive number
Definitions:	The same setpoint assignment has been allocated more than once. MD30110 $MA_CTRLOUT_MODULE_NR contains the same value for different axes.
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Please inform the authorized personnel/service department. Avoid dual assignment of the setpoint by correcting MD30110 $MA_CTRLOUT_MODULE_NR. Also check the selected bus type MD30100 $MA_CTRLOUT_SEGMENT_NR.
Program Continuation:	Switch control OFF - ON.

26019   Axis %1 encoder %2 measurement not possible with this controller module

Parameters:	%1 = NC axis number %2 = Encoder number
Definitions:	If the MD MD13100 $MN_DRIVE_DIAGNOSIS[8] contains a value not equal to zero, then the control has found at least one control module which does not support measuring. Measuring was programmed from the part program for the associated axis.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	If possible, modify the measuring motion such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis. Otherwise, exchange the controller module for one that supports measuring. See MD13100 $MN_DRIVE_DIAGNOSIS[8].
Program Continuation:	Clear alarm with the RESET key. Restart part program

26022	**Axis %1 encoder %2 measurement with simulated encoder not possible**
Parameters:	%1 = NC axis number %2 = Encoder number
Definitions:	Alarm occurs on the control when a measurement was made without the encoder hardware (simulated encoder).
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	- Please inform the authorized personnel/service department. - If possible, modify the measuring motion such that the axis concerned does not have to travel; do not program this axis in the MEAS block again. However, it is then no longer possible to query a measured value for this axis. - Ensure that measurement is not taking place with simulated encoders (MD30240 $MA_ENC_TYPE).
Program Continuation:	Clear alarm with the RESET key. Restart part program

26024	**Axis %1 machine data %2 value changed**
Parameters:	%1 = Axis name, spindle number %2 = String: MD identifier
Definitions:	The machine data contains an invalid value and therefore has been changed by the software.
Reaction:	Alarm display.
Remedy:	Check MD.
Program Continuation:	Clear alarm with the RESET key. Restart part program

26025	**Axis %1 machine data %2[%3] value changed**
Parameters:	%1 = Axis name, spindle number %2 = String: MD identifier %3 = Index: MD array index
Definitions:	The machine data contains an invalid value. It was therefore changed by the software internally to a valid value.
Reaction:	Alarm display.
Remedy:	Check MD.
Program Continuation:	Clear alarm with the RESET key. Restart part program

26030	**Axis %1 encoder %2 absolute position lost**
Parameters:	%1 = Axis name, spindle number %2 = Encoder number
Definitions:	The absolute position of the absolute encoder has become invalid because - on changing parameter block a changed gear stage ratio was identified between encoder and processing or - the encoder has been replaced (the absolute encoder's serial number has changed, see MD34230 $MA_ENC_SERIAL_NUMBER, and drive-specific parameters).
Reaction:	Mode group not ready. The NC switches to follow-up mode. Channel not ready. NC Start disable in this channel. Axes of this channel must be re-referenced. Interface signals are set. Alarm display. NC Stop on alarm. Channel not ready.
Remedy:	Please inform the authorized personnel/service department. Rereferencing/resynchronization of the absolute encoder; attach absolute encoder on the load side and configure correctly (e.g. MD 31040 $MA_ENC_IS_DIRECT).
Program Continuation:	Teileprogramm neu starten.Clear alarm with the RESET key in all channels of this mode group. Restart part program.

26050	**Axis %1 parameter set change from %2 to %3 not possible**
Parameters:	%1 = Axis name, spindle number %2 = Index: current parameter block %3 = Index: new parameter block
Definitions:	The parameter block change cannot be performed without jumps. This is due to the content of the parameter block to be switched on, e.g. different load gear factors.
Reaction:	The NC switches to follow-up mode. Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	In the following cases, the parameter block change is carried out via MD31060 $MA_DRIVE_AX_RATIO_NUMERA and MD31050 $MA_DRIVE_AX_RATIO_DENOM without an alarm, even with different load gear ratio settings: 1. If no position control is active (e.g. in follow-up mode or if spindle is in speed-controlled mode). 2. For position control with the direct encoder. 3. For position control with the indirect encoder (the calculated load position difference must not exceed the value indicated in MD36500 $MA_ENC_CHANGE_TOL).
Program Continuation:	Clear alarm with the RESET key. Restart part program

26051	**[Channel %1:] In block %2 unanticipated stop crossed in continuous path mode**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	The path interpolation did not stop, as required, at the end of the block, but will only decelerate to a standstill in the next block. This error situation occurs if the stop at block change was not planned by the path interpolation or was not detected early enough. A possible cause is that the PLC changed the spindle speed when MD35500 $MA_SPIND_ON_SPEED_AT_IPO_START > 0, and the machine has to wait until the spindle has returned to the setpoint range. Another possible cause is that a synchronized action needs to be finished before the path interpolation continues. The alarm is only output if MD11400 $MN_TRACE_SELECT = 'H400'. The alarm output is normally suppressed. - MD11400 $MN_TRACE_SELECT has SIEMENS password protection.
Reaction:	Alarm display.
Remedy:	MD35500 $MA_SPIND_ON_SPEED_AT_IPO_START = 1. Program G09 before the alarm output in the block to allow the path interpolation to stop as planned.
Program Continuation:	Clear alarm with the Delete key or NC START.

26052	**[Channel %1:] In block %2: path velocity too high for auxiliary function output**
Parameters:	%1 = Channel number %2 = Block number, label
Definitions:	This alarm usually occurs in a block with auxiliary function output during a movement. In this case, the wait for acknowledgement of the auxiliary function was longer than planned. The alarm also occurs if internal control inconsistencies cause continuous path mode (G64, G641, ...) to be blocked unexpectedly. The path interpolation stops abruptly at the end of the block indicated in the message (regenerative stop). On the next block change, the path continues unless the abrupt stop has caused an error in the position controller (e.g. because MD36400 $MA_CONTOUR_TOL setting was over-sensitive).
Reaction:	Alarm display.
Remedy:	- If the alarm occurred in a block with auxiliary function output during the movement: from SW 5.1 or higher, increase machine MD10110 $MN_PLC_CYCLE_TIME_AVERAGE or - Program G09 in the block indicated in the message to allow the path interpolation to stop as planned.
Program Continuation:	Clear alarm with the Delete key or NC START.

26100	**Axis %1 drive %2 sign of life missing**
Parameters:	%1 = Axis name, spindle number %2 = Drive number
Definitions:	Special case: The output of drive number=0 indicates that a computing timeout occurred on the IPO level (see also alarm 4240)
Reaction:	NC not ready. The NC switches to follow-up mode. Mode group not ready, also effective for single axes. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Restart drive, check drive software.
Program Continuation:	Switch control OFF - ON.

26110	**Independent drive stop/retract triggered**
Definitions:	
Reaction:	NC not ready. The NC switches to follow-up mode. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	
Program Continuation:	Clear alarm with the RESET key. Restart part program

26120	**[Channel %1:] Axis %2 $AA_ESR_ENABLE = 1 but axis should be set to NEUTRAL**
Parameters:	%1 = Channel %2 = Axis, spindle
Definitions:	One axis with ESR configuration and $AA_ESR_ENABLE[Achse] = 1 should be set to NEUTRAL. However, neutral axes (apart from single axes) cannot execute an ESR.
Reaction:	Alarm display.
Remedy:	Set $AA_ESR_ENABLE[Achse] = 0 before setting axis to NEUTRAL. Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.
Program Continuation:	Clear alarm with the Delete key or NC START.

26121	**[Channel %1:] Axis %2 is NEUTRAL and $AA_ESR_ENABLE = 1 should be set**
Parameters:	%1 = Channel %2 = Axis, spindle
Definitions:	$AA_ESR_ENABLE[Achse] = 1 should not be set to neutral axes (apart from single axes). Neutral axes (apart from single axes) cannot execute an ESR.
Reaction:	Alarm display.
Remedy:	Do not apply $AA_ESR_ENABLE[Achse] = 1 to neutral axes (apart from single axes). Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.
Program Continuation:	Clear alarm with the Delete key or NC START.

26122	**[Channel %1:] Axis %2, $AA_ESR_ENABLE = 1, axis exchange not executed in this state**
Parameters:	%1 = Channel %2 = Axis, spindle

Definitions:	With $AA_ESR_ENABLE[Achse] = 1 axis exchange not permitted.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm.
Remedy:	Set $AA_ESR_ENABLE[axis] = 0 before axis exchange.
Program Continuation:	Clear alarm with the RESET key. Restart part program Set $AA_ESR_ENABLE[axis] = 0

26124 [Channel %1:] Axis %2, $AC_ESR_TRIGGER triggered but axis is NEUTRAL and cannot execute ESR

Parameters:	%1 = Channel %2 = Axis, spindle
Definitions:	Channel-specific ESR ($AC_ESR_TRIGGER) triggered, but one axis with ESR configuration is NEUTRAL at the time of triggering. Neutral axes are ignored with ESR (apart from single axes which react only to $AA_ESR_TRIGGER[Ax]).
Reaction:	Alarm display.
Remedy:	$AA_ESR_ENABLE[Achse] = 1 should not be set with neutral axes. Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.
Program Continuation:	Clear alarm with the Delete key or NC START.

26126 [channel %1:] block %2 axis %3: ESRR or ESRS could not be executed, error code %4

Parameters:	%1 = Channel %2 = Block number, label %3 = Axis, spindle %4 = Error code
Definitions:	Writing the data of the part program commands ESRR or ESRS could not be executed. Error code: 1: The indicated axis is not assigned to any axis. 2: One or more ESR parameters in SINAMICS are not available. 3: Writing of one or more ESR parameters in SINAMICS was prevented.
Reaction:	Local alarm reaction. NC Start disable in this channel. Interface signals are set. Alarm display. NC Stop on alarm at block end.

Remedy:	Error code:
	1: Check the assignment of the NC axes to the drives.
	2: One or more ESR parameters in SINAMICS are not available.
	  Check the assignment of the NC axes to the drives.
	  Programming of ESRR or ESRS is possible only for SINAMICS as from V4.4.
	  The function module "drive-autonomous stop and retract" in SINAMICS not active.
	3: Writing one or more ESR parameters in SINAMICS was prevented.
	  Activate the output of additional information alarms through MD11411 $MN_ENABLE_ALARM_MASK, bit 1 = 1.

Program Continuation:	Clear alarm with the RESET key. Restart part program Writing of the data of the part program commands ESRR or ESRS was refused. Check programming Acknowledge alarm with reset

3.3 PLC alarms

400000 PLC STOP %1

Parameters:	%1 = Error type
Definitions:	PLC not in cyclic mode. Travel with the machine is not possible.
	%1:1 Ready(User program has not been started)
	%1:2 Break (User program has been interrupted)
	%1:3 Error (Other PLC alarm with PLC Stop active)
Reaction:	Alarm display.
Remedy:	Rectify other PLC alarm;
	set PLC stop from Startup menu
	or test user program.
Program Continuation:	Alarm display showing cause of alarm disappears. No further operator action necessary.

400001 System error %2

Parameters:	%2 = Error type
Definitions:	With this alarm, internal alarm states are displayed that, in conjunction with the transferred error text, provide information about the cause and location of the error.
Reaction:	PLC Stop
Remedy:	Notify Siemens of this error together with the error message.
Program Continuation:	Switch control OFF - ON.

400002 System error %1

Parameters:	%1 = Type number
Definitions:	With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.
Reaction:	PLC Stop
Remedy:	Report this error to Siemens along with the type number.
Program Continuation:	Switch control OFF - ON.

400003 System error, connection to MCP canceled

Definitions:	This alarm reports the loss of communications to the MCP.
Reaction:	PLC Stop
Remedy:	Check MCP connection.
Program Continuation:	Clear alarm with the Delete key or NC START.

400004 Code error: %2 network %1

Parameters:	%1 = Network number
	%2 = Internal error code, block type
Definitions:	The user program contains an operation not supported by the controller.
Reaction:	PLC Stop
Remedy:	Modify and reload user program.
Program Continuation:	Switch control OFF - ON.

400005 PLC stop specified from start-up menu

Definitions:	User program is not being processed
Reaction:	Alarm display.

Remedy:	Switch controller off/on
Program Continuation:	Switch control OFF - ON.

400006   **Loss of remanent PLC data**

Definitions:	The following causes are possible: Control handling (e.g. standard PLC deletion, power up with default values) Control handling of power up with backed up data without backing up data in advance Support time exceeded
Reaction:	Alarm display.
Remedy:	Update the data required.
Program Continuation:	Clear alarm with the Delete key or NC START.

400007   **Operand error: %3 %2 network %1**

Parameters:	%1 = Network number %2 = Block type, block number %3 = Variable address
Definitions:	
Reaction:	PLC Stop
Remedy:	The variable displayed must be checked in the user program for violation of the address range, impermissible data type and alignment errors.
Program Continuation:	Switch control OFF - ON.

400008   **Programming tool - version is not compatible %1 %2**

Parameters:	%1 = Programming tool version
Definitions:	This version is not compatible with the product level of the controller.
Reaction:	PLC Stop
Remedy:	Translate the user program using a suitable programming tool version and load in the control.
Program Continuation:	Switch control OFF - ON.

400009   **Computing time overrun at PLC level: %2 network %1**

Parameters:	%1 = Network number %2 = Block type
Definitions:	Check user program of the corresponding displayed network.
Reaction:	PLC Stop
Remedy:	Change user program
Program Continuation:	Switch control OFF - ON.

400010   **Arithmetic error in user program: Type %2 network %1**

Parameters:	%1 = Network number, block ID %2 = Error type
Definitions:	Check user program in the stated network. %2 = 1:Division by zero in fixed point arithmetic %2 = 2:Floating-point arithmetic
Reaction:	PLC Stop
Remedy:	Change user program.
Program Continuation:	Switch control OFF - ON.

400011   **Maximum number of subroutine levels exceeded: %2 network %1**

Parameters:	%1 = Network number %2 = Block ID
Definitions:	Check user program in the stated network.
Reaction:	PLC Stop
Remedy:	Change user program.
Program Continuation:	Switch control OFF - ON.

400012   **Error affecting indirect addressing: %2, network %1**

Parameters:	%1 = Network number %2 = Block ID
Definitions:	Check user program in the stated network.
Reaction:	PLC Stop
Remedy:	Change user program.
Program Continuation:	Switch control OFF - ON.

400013   **PLC user program is incorrect**

Definitions:	The PLC user program in the control is incorrect or is not available.
Reaction:	PLC Stop
Remedy:	Reload PLC user program.
Program Continuation:	Switch control OFF - ON.

400017   **PLC TOOLMAN: missing table in DB9900**

Definitions:	The PLC TOOLMAN cannot find one of the tables 9900, 9901 or 9902.
Reaction:	PLC Stop
Remedy:	Create the missing table(s).
Program Continuation:	Switch control OFF - ON.

400018   **PLC TOOLMAN: Spindle or load location invalid**

Definitions:	The PLC TOOLMAN does not know the specified spindle or load location number.
Reaction:	PLC Stop
Remedy:	Specify the correct number.
Program Continuation:	Switch control OFF - ON.

400019   **PLC maintenance planner: Error in DB 9903 or 9904**

Definitions:	PLC maintenance planner: DBs 9903 and 9904 must be present and must have the same length.
Reaction:	PLC Stop
Remedy:	Create block(s) correctly.
Program Continuation:	Switch control OFF - ON.

400020   **PLC mainentance request %1 : Initial data not plausible**

Definitions:	Redefine the initial data in accordance with the documentation. Error causes: Too many alarms for the interval length Time of first alarm too late

	Time of first alarm greater than interval
Reaction:	Alarm display.
Remedy:	Specify correct initital data according to the documentation.
Program Continuation:	Clear alarm with the Delete key or NC START.

400022   The PLC functionality: %2 is not permitted.

Definitions:	PLC functionality: %2 The specified PLC functionality is not supported.
Reaction:	Alarm display.
Remedy:	You are not permitted to use the specified PLC functionality.
Program Continuation:	Clear alarm with the Delete key or NC START.

400023   Arithmetic problem in UP (REAL-Operation) OB/SBR/INT %1 Netzwerk %2

Parameters:	%1 = Block no. %2 = Network no.
Definitions:	Problems with REAL operands occurred during execution of the PLC user program
Reaction:	Alarm display.
Remedy:	The variables used have to be checked for valid values.
Program Continuation:	Clear alarm with the Delete key or NC START.

400024   Dynamically managed memory used up, area %1

Definitions:	Memory overflow in area corresponding to memory area identification %1= 1xx: dynamic RAM, xx refers to internal RAM class %1= 2: MMF (user project) %1= 3: buffered RAM %1= 4: UP-RAM
Reaction:	Alarm display.
Remedy:	With %1 = 1xx: Internal error due to too many external communications requirements With %1 > 1: User project too large or too many operations
Program Continuation:	Clear alarm with the Delete key or NC START.

400025   PLC Ctrl Energy: Error in DB 9906

Definitions:	PLC Ctrl Energy: DB 9906 has been deleted.
Reaction:	PLC Stop
Remedy:	DB 9906 of the programming tool library must be incorporated into the project.
Program Continuation:	Switch control OFF - ON.

3.4   PLC user alarms

700000 to 700127	PLC User alarms
For the default PLC program, refer to descriptions of PLC subroutines in the *Commissioning Manual* for detailed information about the PLC user alarms.	
For an OEM PLC program, refer to relevant descriptions of machine tool for detailed information about the PLC user alarms	

System responses

4

4.1 System reactions to SINUMERIK alarms

Identifier	COMPBLOCKWITHREORG
Effects	Block preparation has detected an error, which can be rectified by modifying the program. Reorganization is performed after a program modification. • Correction block with reorganization.

Identifier	COMPENSATIONBLOCK
Effects	Block preparation has detected an error, which can be rectified by modifying the program. • Correction block.

Identifier	FOLLOWUP
Effects	Follow-up of axes. • NC switches to follow-up mode.

Identifier	INTERPRETERSTOP
Effects	Program execution is aborted after all the prepared blocks (IPO buffer) have been processed. • Interpreter stop.

Identifier	LOCALREACTION
Effects	• Local alarm reaction.

Identifier	NOALARMREACTION
Effects	• No alarm reaction.

Identifier	NOREADY | NCKREACTIONVIEW
Effects	NCK ready off: Active fast braking (i.e. with maximum braking current) of all drives, the controller enable for all NC axes is deleted, the NC ready relay drops out. • NC not ready.

Identifier	NOREADY | BAGREACTIONVIEW
Effects	Mode group ready off: Active fast braking (i.e. with maximum braking current) of the drives in this mode group, the controller enable of the NC axes involved is deleted. • Mode group not ready.

Identifier	NOREADY
Effects	Channel ready off: Active fast braking (i.e. with maximum braking current) of the drives in this channel, the controller enable of the NC axes involved is deleted. • Channel not ready.

Identifier	NONCSTART
Effects	It is not possible to start a program in this channel. • NC start inhibit in this channel.

Identifier	NOREFMARK
Effects	The axes in this channel have to be referenced again. • Re-reference axes in this channel.

Identifier	SETVDI
Effects	VDI interface signal alarm is set. • Interface signals are set.

Identifier	SHOWALARM
Effects	Alarm is displayed on the HMI. • Alarm display.

Identifier	STOPBYALARM
Effects	Ramp stop of all channel axes. • NC stop for alarm.

Identifier	STOPATENDBYALARM
Effects	Stop at end of block. • NC Stop for alarm at end of block.

Identifier	SHOWALARMAUTO
Effects	The alarm is displayed whenever bit 0 of machine data ENABLE_ALARM_MASK is set. The reaction should be set whenever an alarm should only occur during automatic mode without manual operation by the user. • Alarm reaction in automatic mode

Identifier	SHOWWARNING
Effects	The alarm is displayed whenever bit 1 of machine data ENABLE_ALARM_MASK is set. It is used for alarms which should normally be suppressed. • Message display.

Identifier	ALLBAGS_NOREADY
Effects	The Ready is canceled in all mode groups. The reaction thus corresponds to an NCKREACTIONVIEW | NOREADY, the difference being that the NC READY relay is not canceled and the corresponding VDI bit is not set. This is desirable in the event of an emergency stop for example. • Mode group not ready.

Identifier	DELAY_ALARM_REACTION
Effects	If this alarm reaction is configured in the alarm handler, all alarm reactions for alarms, which occur at this point, are buffered channel-specifically and are, therefore, not active. The alarms are displayed on the HMI. Mode group and NCK-wide reactions are transferred. The reaction is cleared by activating the clearDelayReaction call or by an alarm, which has configured NO_DELAY_ALARM_REACTION. This activates all the delayed alarm reactions. • All channel-specific alarm reactions delayed on alarm, alarm display.

Identifier	NO_DELAY_ALARM_REACTION
Effects	The DELAY_ALARM_REACTION state is canceled. • The alarm reaction delay is canceled.

Identifier	ONE_IPO_CLOCK_DELAY_ALARM_REACTION
Effects	All alarm reactions are delayed by one cycle when an alarm is output. This functionality became necessary as part of ESR development. • All alarm reactions are delayed by one IPO cycle on alarm.

4.2 Cancel criteria for alarms

Identifier	CANCELCLEAR
Effects	The alarm is cleared in any channel when the Cancel key is pressed. It is also cleared by the Start part program key. • Clear the alarm with the **"ALARM CANCEL"** key or with the **"CYCLE START"** key.

Identifier	CLEARHIMSELF
Effects	Self-clearing alarm. The alarm is not cleared by an operator action but explicitly by a "clearAlarm" programmed in the NCK source code. • The alarm is no longer displayed when the alarm cause has been removed. No other operator actions are required.

Identifier	NCSTARTCLEAR
Effects	The alarm is cleared by starting a program in the channel, in which the alarm occurred. The alarm is also cleared by an NC reset. • Clear the alarm with the **"CYCLE START"** key or the RESET key and continue the program.

Identifier	POWERONCLEAR
Effects	The alarm is canceled by switching off the control and switching it on again. • Switch the control OFF - ON.

Identifier	RESETCLEAR
Effects	The alarm is cleared by pressing the Reset key in the channel in which the alarm occurred. • Clear the alarm with the RESET key. Restart the part program.

Identifier	BAGRESETCLEAR
Effects	The alarm is cleared by a "BAGRESETCLEAR" command or by carrying out a reset in all channels of this mode group. • Press the RESET key to clear the alarm in all channels of this mode group. Restart the part program.

Identifier	NCKRESETCLEAR
Effects	The alarm is cleared by an "NCKRESETCLEAR" command or by carrying out a reset in all channels. • Clear alarm in all channels with the RESET key. Restart the part program.

Identifier	NOCLEAR
Effects	The clear information is only required for the internal pseudo alarm number EXBSAL_NOMOREALARMS.

SINAMICS V60 alarms

5

5.1 Overview of alarms

Table 5-1   Overview of alarms

Alarm code	Alarm name	Description
A01	Power board ID number error	Unable to identify the power board
A02	Parameter error	Parameter validation error (CRC error, encoder type or parameter header invalid)
A03	Memory-write failure	Unable to write data to memory
A04	Control voltage error	Control voltage is lower than 3.5 V.
A05	IGBT overcurrent	IGBT is detected overcurrent.
A06	Internal chip overcurrent	Internal chip is detected overcurrent.
A07	Grounding short circuit	Grouding short circuit occurs during drive initialization
A08	Endoder UVW sginals error	Signals from encoder phases U, V, W are detected all the same (all high or all low)
A09	Encoder TTL signals error	TTL pulse error
A14	Internal error	Software failure
A21	DC link overvoltage	DC link voltage is higher than 405 V
A22	IT protection	IGBT current exceeds the upper limit for 300 ms
A23	DC link undervoltage	DC link voltage is lower than 200 V
A41	Overspeed	Actual motor speed is higher than 2200 rpm
A42	IGBT overtemperature	IGBT becomes overheating
A43	Over-position	Position offset is out of limit
A44	I^2T protection	Motor load exceeds nominal motor torque
A45	Emergency stop	Enable signal from Terminal 65 is lost during normal drive operation

5.2 Alarm list

Alarm code	Background	Possible Cause	Remedy	Result	Acknowledgement
A01		Power board is broken	Replace the drive with a new one	Free stop	Power on
A02		The memory is damaged due to unexpected power-off during data saving	Restore default parameters	Free stop	Power on
A03		Memory is damaged	Replace the drive with a new one	Free stop	Power on

Alarm code	Background	Possible Cause	Remedy	Result	Acknowledgement
A04		The 24 V DC supply is abnormal	Check the 24 V DC supply	Free stop	Power on
		The drive is defective	Replace the drive with a new one		
A05	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	1. There is a short-circuit between terminals U, V, W and PE on the drive 2. bad grounding 3. motor insulation is broken 4. the drive is broken	1. check the wiring 2. make a correct grounding 3. replace the motor with a new one 4. replace the drive with a new one	Free stop	Power on
A06	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	1. There is a short-circuit between terminals U, V, W and PE on the drive 2. bad grounding 3. motor insulation is broken the drive is broken	1. check the wiring or whether the connection to U, V or W is broken 2. make a correct grounding 3. replace the motor with a new one 4. replace the drive with a new one	Free stop	Power on
A07	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	1. IGBT module is broken 2. There is a short-circuit between U, V or W and PE	1. replace the drive with a new one 2. Check the wiring	Free stop	Power on
A08		1. UVW signals of the encoder are bad 2. bad cable 3. bad cable shielding 4. bad wiring of the shielded ground cable 5. there is a failure in the interface circuit of the encoder	1. replace the encoder with a new one 2. check the interface circuit of the encoder	Free stop	Power on
A09		1. connection failure of encoder ABZ 2. bad cable 3. bad cable shielding 4. bad wiring of the shielded grounded cable 5. there is a failure in the interface circuit of the encoder	1. check the wiring of encoder cable 2. check the interface circuit of the encoder	Free stop	Power on

Alarm code	Background	Possible Cause	Remedy	Result	Acknowledgement
A14		There is failure at the internal software	Reset by power-on	Free stop	Power on
		A short-circuit occurs to the encoder	Check the wiring of encoder		
A21	This alarm may occur when the 24 V DC supply is connected	There is a failure at the circuit board	Replace the drive with a new one	Free stop	Press Enter key on the operator panel or terminal RST of X6 interface
	This alarm may occur when the DC link is connected	1. the mains supply voltage is too high 2. the waveform of mains supply voltage is abnormal	Check the power supply		
	This alarm may occur when the motor is running	1. disconnection of the internal brake resistor 2. the internal brake resistor is broken	Replace the drive with a new one		
		Brake loop has no enough space	1. lower the start-stop frequency 2. reduce the limit value of current 3. reduce load inertia 4. use another drive and motor with higher frequencies		
A22		The motor is mechanically blocked	Check the mechanical load	Free stop	Press Enter key on the operator panel or terminal RST of X6 interface
		Overload	1. lighten the load 2. use another drive and motor with higher frequencies		
A23		1. circuit board fails 2. fuse of the power is burnt out 3. rectifier is broken	Replace the drive with a new one	Free stop	Press Enter key on the operator panel or terminal RST of X6 interface
		1. low supply voltage 2. insufficient supply power capacity 3. transient power failure	Check the power supply		

Alarm code	Background	Possible Cause	Remedy	Result	Acknowledgement
A41	This alarm may occur when the 24 V DC supply is connected	Circuit board fails	Replace the drive with a new one	Emergency stop (the motor will stop with the maximum energy or torque)	Press Enter key on the operator panel or terminal RST of X6 interface
		The encoder fails	Replace the motor with a new one		
	This alarm may occur when the motor is running	The encoder fails	Replace the drive with a new one		
		The encoder cable is badly connected	Replace the motor with a new one		
	This alarm may occur when the motor starts running	1. terminals U, V and W on the motor are wrongly connected 2. the encoder is wrongly wired	Check the wiring		
A42		Ambient temperature is too high	Check the ambient temperature	Emergency stop (the motor will stop with the maximum energy or torque)	Press Enter key on the operator panel or terminal RST of X6 interface
		The drive is overloaded	Check the drive load		
		Circuit board fails	Replace the drive with a new one		
A43	This alarm may occur when the 24 V DC supply is connected	Circuit board fails	Replace the drive with a new one	Emergency stop (the motor will stop with the maximum energy or torque)	Press Enter key on the operator panel or terminal RST of X6 interface
	The motor does not rotate or reversely rotate if the command pulse is input after DC-link and mains line are connected	1. terminals U, V and W on the motor is wrongly connected 2. encoder cable is wrongly connected	Check the wiring		
		Encoder fails	Replace the motor with a new one		
	This alarm may occur when the motor is running	The permitted maximum position difference is too small	Set a wider value range for the detection of position over-tolerance (P34)		
		The position loop gain is too small	Give more gains		
		No enough torque	Check the limit value of current		
			Reduce the load		
			Use a drive and motor with higher frequencies		
		Low speed	Check the maximum speed limitation (refer to parameter P26)		
		Command pulse frequency is too high	1. Lower the frequency 2. Check whether P36 has the right value		

Alarm code	Background	Possible Cause	Remedy	Result	Acknowledgement
A44	This alarm may occur when the 24 V DC supply is connected	The circuit board fails	Replace the drive with a new one	Emergency stop (the motor will stop with the maximum energy or torque)	Press Enter key on the operator panel or terminal RST of X6 interface
	This alarm may occur when the DC link is connected	The rated torque is exceeded	1. check the load 2. lower the start/stop frequency 3. use drive and motor with more power		
		The brake is not open	Check whether the brake is open or not		
		The motor is not stable	1. modify the gain value 2. lessen load inertia		
		The encoder is wrongly wired	Check the wiring		
A45		The 65 enable signal is lost when the motor is running	Check the 65 enable terminal	Emergency stop (the motor will stop with the maximum energy or torque)	Press Enter key on the operator panel or terminal RST of X6 interface

Note

Alarms with alarm code < A21 can be cancelled by power-on, while alarms with alarm code ≥ A21 can be cancelled by RST terminal.

5.3 Errors during drive self-test

The drive module always conducts a self-test at every power-on. If any error occurs during this period, the drive screen form will shown one of the following error codes:

Drive display	Description	Cause	Remedy
E1	Error 1	RAM damaged	Replace the drive
E2	Error 2	Flash damaged	Replace the drive
E3	Error 3	Program copy error	Replace the drive

5.4 Other faults

1. Motor speed incorrect

   – Description: The actual motor position is always wrong, and the motor speed is not right, either.

   – Cause: The pulse signal from the NC to drive is incorrect.

   – Remedy: Check the pulse signal line connection

2. Brake not open

   – Description: The brake is not open when the drive is in "S-Run" state.

   – Cause: A short circuit has occurred in the brake cable.

   – Remedy: Check brake cable connection

Data backup   6

6.1   Overview of internal/external data backup

You can back up user data internally or externally on the control system.

Note

Archiving/data backup

It is recommended that you regularly back up the internal SINUMERIK memory on a USB
stick. You can transfer the backed up data to the SINUMERIK later on. In this way you can
restore the previous status of the unit.

You must perform an external data backup when you have made major data changes or after
you have commissioned the control system.

6.2   Internal data backup

You can save the data of the limited-buffered memory via a backup copy to the permanent
memory of the control system. This backup is performed internally and is always necessary if
you need to switch off the control system for longer than 60 hours.

Note

After changing important data, it is recommended to carry out a data backup **immediately**.

Performing an internal data backup

1. In the "SYSTEM" operating area (key combination: [SHIFT] + [SYSTEM ALARM]), press the "Save data" softkey.
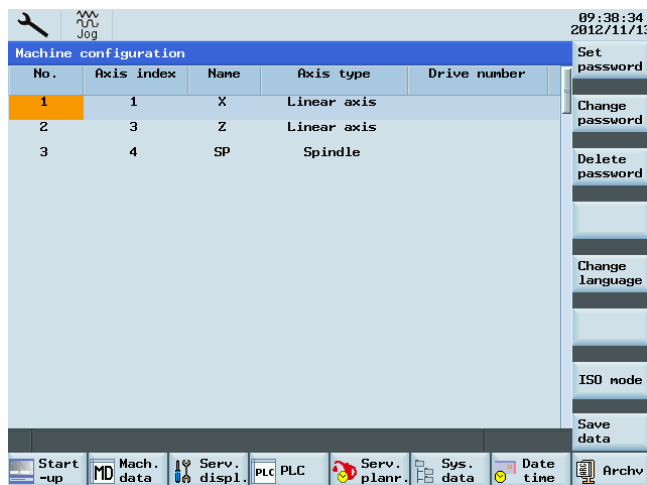
2. Click "OK" to confirm the message that prompts.
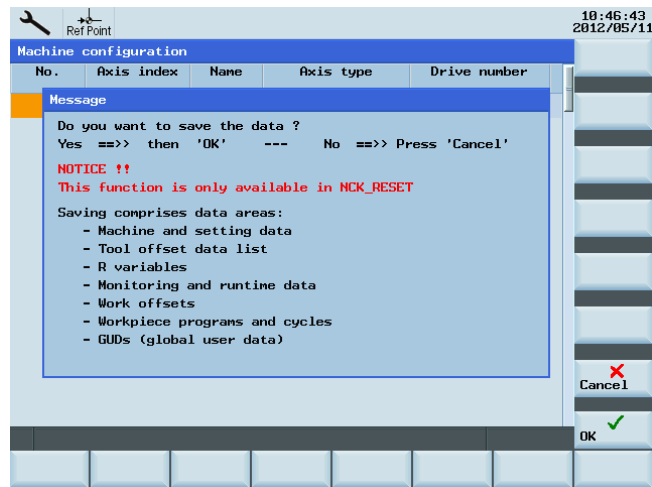
3. The control system begins to save data. Wait until the progress dialog disappears and the data saving finishes.
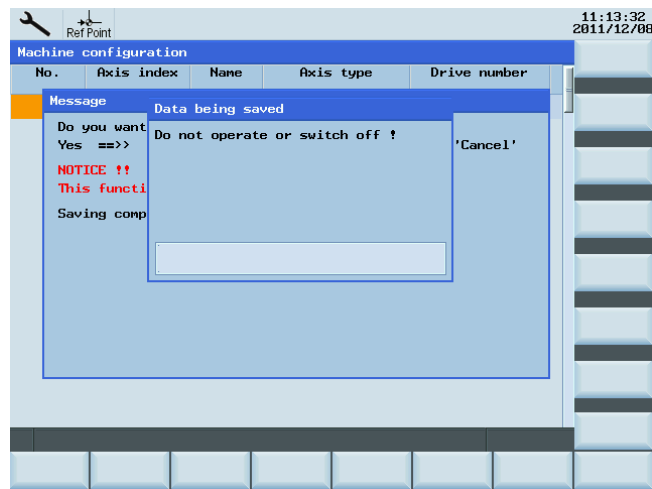
Note

While making an internal data backup, you must neither operate nor turn off the control system.

Loading internally backed-up data

To load the internally backed-up data, proceed as follows:

1. In the "SYSTEM" operating area, press the "Start-up" softkey.

2. In the "NC start-up" window, select the option "Power-up with saved data":
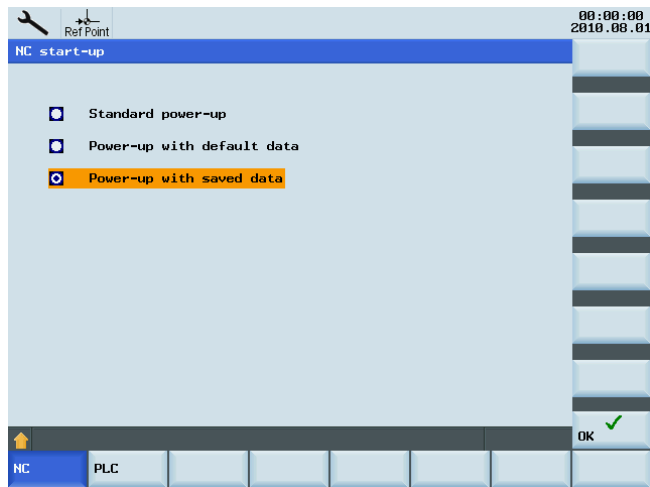
3. Press "OK", and the NC automatically restarts with the saved data.

Note

Message "4062 Backup data loaded" is displayed on the screen.

You must enter the password again after you have powered up the control system with the saved data.

6.3 External data backup

6.3.1 External data backup in a data archive

1. In the "SYSTEM" operating area (key combination: [SHIFT] + [SYSTEM ALARM]), press the "Archv." softkey to open the "Start-up archive" window.
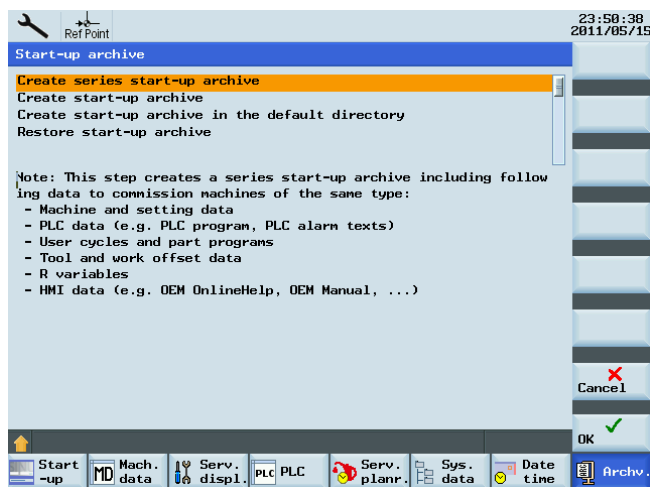
There are three options for creating a data archive:

Create series start-up archive: you can use this option to create a data archive for series machine commissioning, and it contains the following data:

– Machine and setting data
– PLC data (e.g. PLC program, PLC alarm texts)
– User cycles and part programs
– Tool and work offset data
– R variables
– HMI data (e.g. OEM Online Help, OEM Manual, ...)

Create start-up archive: you can use this option to create a data archive for a complete system backup, and it contains the following data:

– Compensation data
– Machine and setting data
– PLC data (e.g. PLC program, PLC alarm texts)
– User cycles and part programs
– Tool and work offset data
– R variables
– HMI data (e.g. OEM Online Help, OEM Manual, ...)

Create start-up archive in the default directory: you can use this option to back up the complete system directly on the system CompactFlash Card (CF card), and it contains the following data:

– Compensation data
– Machine and setting data
– PLC data (e.g. PLC program, PLC alarm texts)
– User cycles and part programs
– Tool and work offset data
– R variables
– HMI data (e.g. OEM Online Help, OEM Manual, ...)

The third option backs up exactly the same data with the option "Create start-up archive". In case of service (e.g. hardware replacement), it can be more convenient to restore the control system directly from the system CF card than from other paths. After creating a start-up archive in the default directory, the option **"Restore start-up archive from the default directory"** appears on the screen:
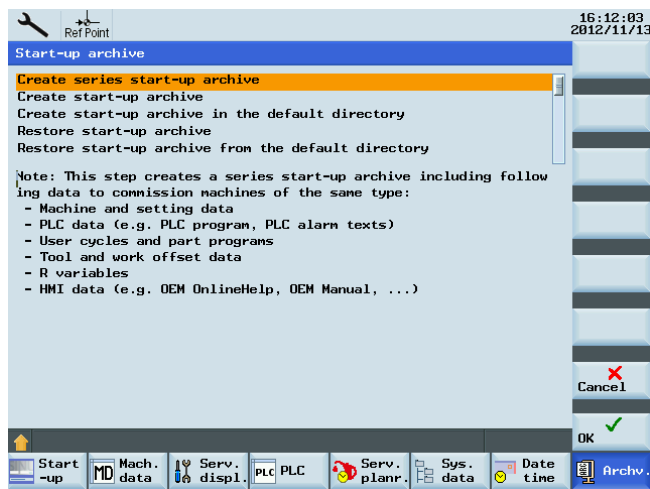
2. Select an option to create your desired data archive for backup and press the "OK" softkey to confirm.

   If you choose to create a **series start-up archive** or a **start-up archive**, you must specify a backup path for the data archive:

   – OEM files: folder on the control system for storing OEM files

   – User files: folder on the control system for storing user files

   – USB: USB stick

   If you choose to create a **start-up archive in the default directory**, it is not necessary to specify a backup path since the archive is automatically saved on the system CF card.

   The following takes **series start-up archive** as an example, and the name of the data archive is "arc_product.arc" by default. You can use your favourite name for it. Select your desired folder and press the <INPUT> key to open it.
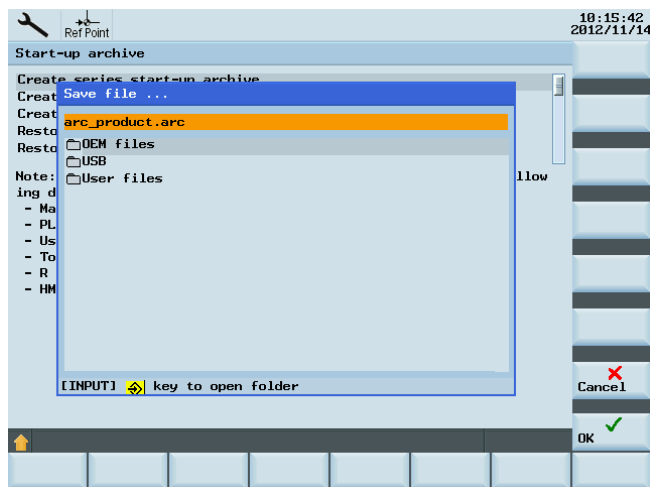
3. Press the "OK" softkey to confirm you selection.
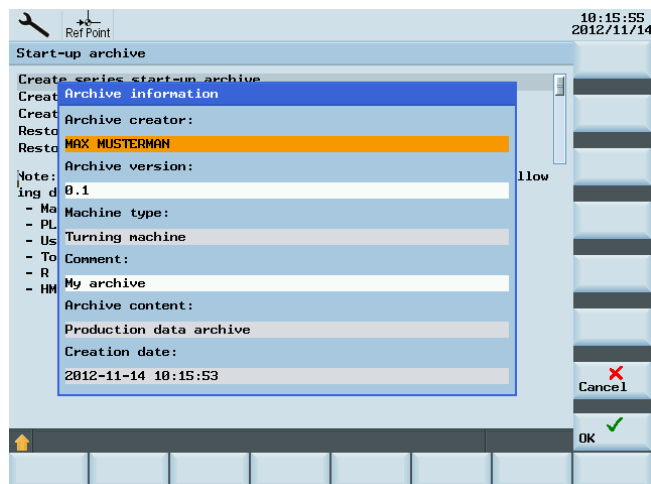
In the "Archive information" box that follows, you can enter the information below:

– Archive creator

– Archive version

– Comment

4. Press the "OK" softkey. The control system starts creation of the start-up archive.

Note

Do not remove the USB stick in the process of data back-up.

6.3.2 External data backup of files

1. When you are in the "SYSTEM" operating area (key combination: SHIFT + SYSTEM ALARM), press the horizontal "Sys. data" softkey to open the following window:
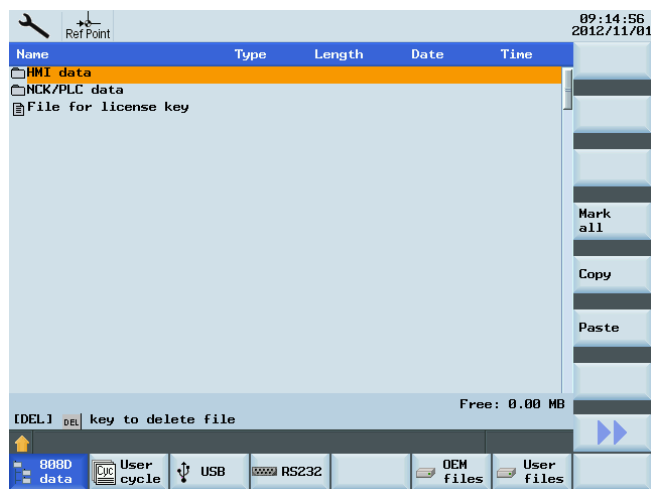
2. Two folders and one "File for license key" are available in the "808D data" window. Select a folder and press <INPUT> to open it.

- ● "HMI data" folder, which includes the following sub-folders:

  – Customized bitmaps
  – User cycle files
  – EasyXLanguage scripts
  – OEM online help (*.txt; *.png; *.bmp)
  – Extended user text file (almc....txt)
  – OEM MD description file (md_descr....txt)
  – OEM manual (oemmanual.pdf)
  – PLC alarm texts (alcu....txt)
  – OEM slideshow (*.bmp; *.png)
  – OEM R variable name file (rparam_name....txt)
  – Service planner task name file (svc_tasks....txt)

- ● "NCK/PLC data" folder, which includes the following data:

  – Leadscrew error compensation
  – Global user data
  – Machine data
  – OEM PLC application (*.pte)
  – R variables
  – Setting data
  – Tool data
  – Work offset

3. Select files that you desire to back-up, and press "Copy". To copy all files in the folder, press "Mark all" and then press "Copy".

4. Select a data back-up path by pressing one of the following softkeys:

   – User cycle: a folder on the control system for storing user cycles
   – USB: USB stick
   – RS232: a PC/PG connected via the RS232 interface
   – OEM files: a folder on the control system for storing OEM files
   – User files: a folder on the control system for storing user files

5. Press "Paste" to complete the data back-up.

Note

Do not connect or disconnect the RS232 cable when the PPU is connected to the line supply.

The settings of the RS232 interface of the 808D and the COM interface on the PG/PC must be identical.

Do not remove the USB stick during the back-up process.

6.3.3 External data backup in case of backlight failure

In case of failure of the backlight of the control system, menu-assisted operation is no longer possible.

To back up data externally in this situation, proceed as follows:

1. Insert a USB stick.

2. Switch the control system on.

3. Wait until the control system has booted.

   The LEDs (RDY: green; POK: green; TEMP: dark) on the PPU indicate the status "Ready for operation".

4. Press the <CTRL> and <S> keys. The external data back-up starts.

5. Wait for about 10 minutes.

   The series archive (NC/PLC/HMI) is exported with the most recent data to the USB stick with the name "808dibn.arc".

6.3.4 External data backup through the RS232 interface

The following data on the NC controller can be read out in the text format to a PC/PG with an RS232 cable:

- NCK/PLC data

- HMI data

- User cycles

- OEM files

- User files

Communication tool - SinuComPCIN

To enable the RS232 communication between a SINUMERIK 808D and a PC/PG, you must have the RS232 communication tool SinuComPCIN installed on your PC/PG. You can get this tool from the SINUMERIK 808D Toolbox.

Data transferring with an RS232 cable and relevant settings

Proceed as follows to transfer data with an RS232 cable:

1. Connect the control system with the PC/PG using an RS232 cable.

2. Configure the communication setttings on the PPU:

   In the "SYSTEM" operating area, press "Sys. data" > "RS232" > "Settings" to open the "Communication settings" window:

Press the <SELECT> key to toggle between different settings. Press "Save" to save the settings. Press "Back" to return to the RS232 main screen.

3. Configure SinuComPCIN on your PC/PG:

Press the "RS232 Config" button and select the desired baudrate from the list:
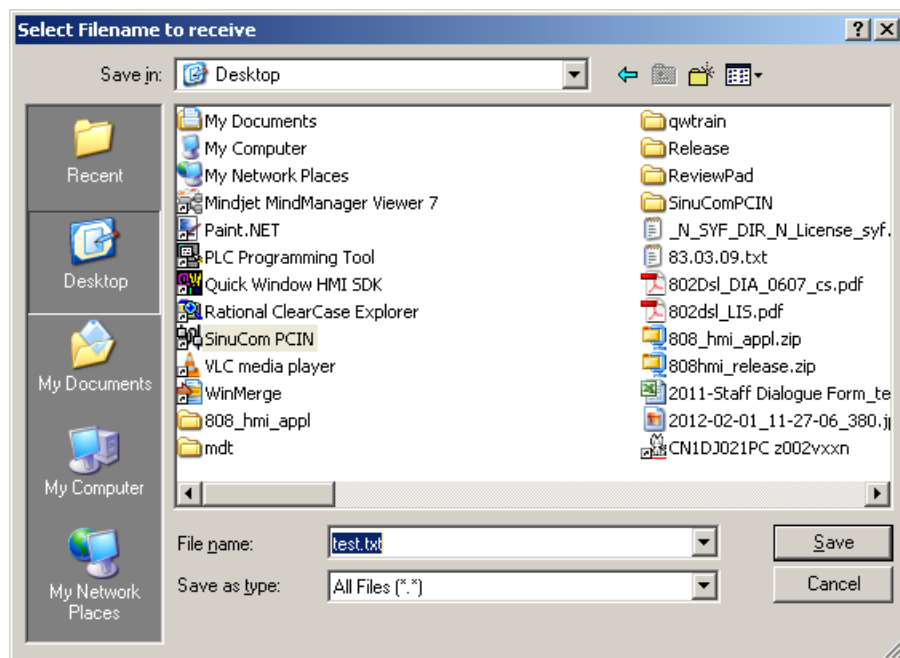

Save the settings with the "Save" button.

Note

This baudrate must be the same as that you have selected on the NC side.

4. Press the "Back" button to return to the main screen of SinuComPCIN:
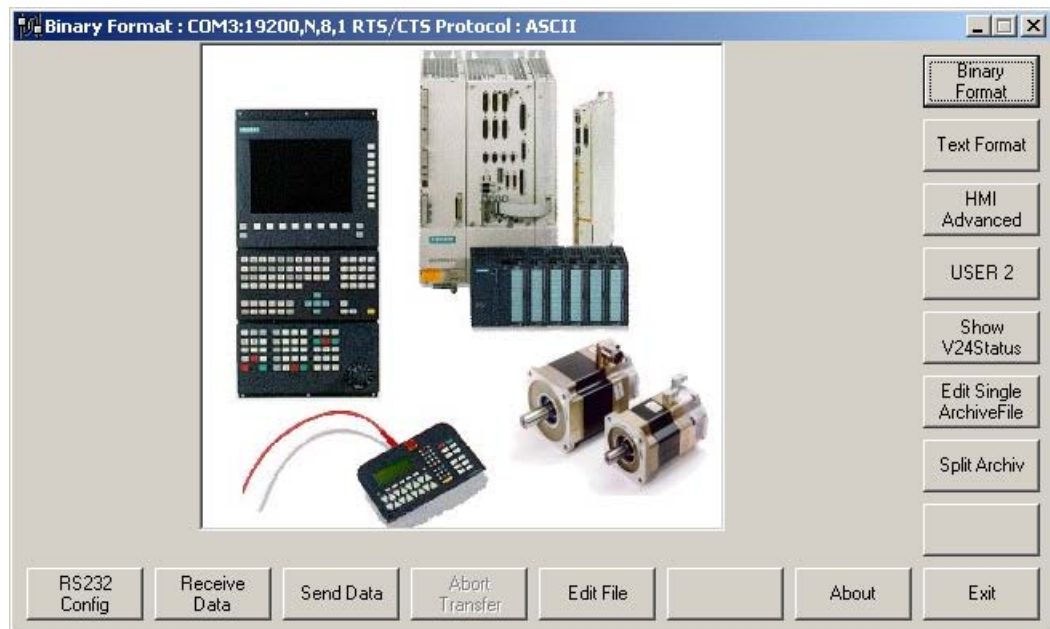
5. Press the "Receive Data" button. Input the name for the text file. For example, test.txt:
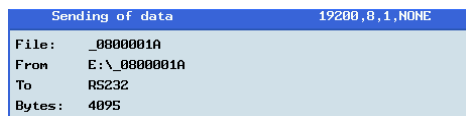

After that press the "Save" button.

6. Select an NC file (take the User cycle CYCLE999.SPF for an example) and copy it with the "Copy" softkey.

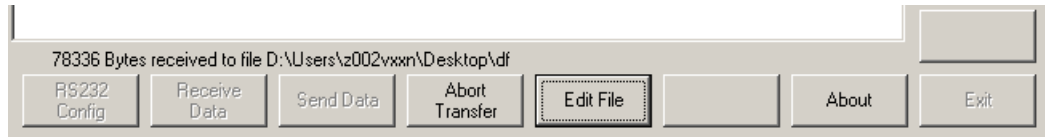7. Switch to the RS232 screen with the "RS232" softkey. Press the vertical "Send" softkey.

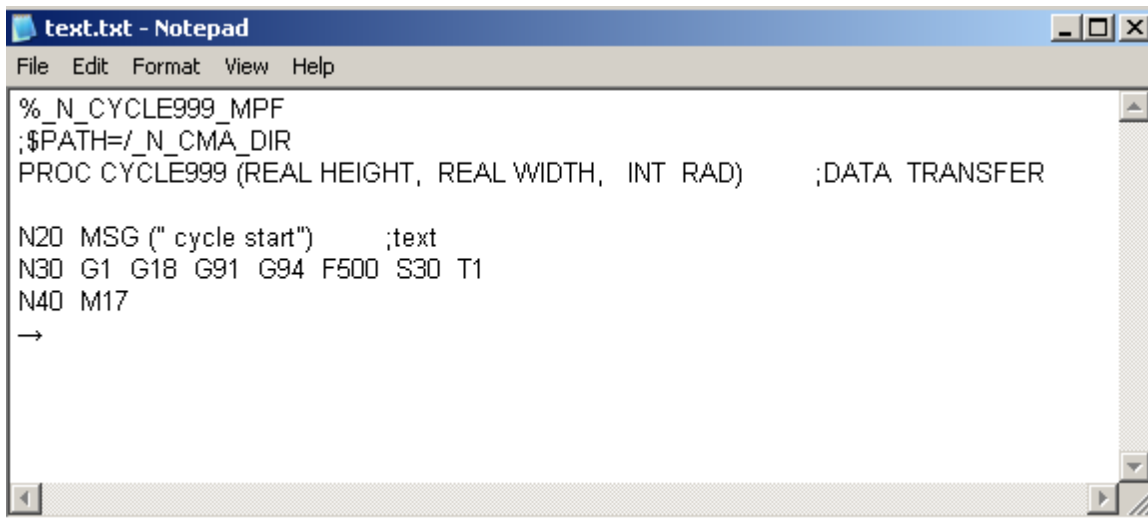8. Under data transferring.

NC side:

```
Sending of data                19200,8,1,NONE
File:     _0800001A
From      E:\_0800001A
To        RS232
Bytes:    4095
```

SinuComPCIN:

78336 Bytes received to file D:\Users\z002vxxn\Desktop\df

RS232 Config | Receive Data | Send Data | Abort Transfer | Edit File | | About | Exit

Wait until SinuComPCIN has finished receiving NC data, and then click the "Abort Transfer" button.

94673 Bytes received to file D:\Users\z002vxxn\Desktop\gfg.txt

RS232 Config | Receive Data | Send Data | Abort Transfer | Edit File | | About | Exit

9. Once the SinuComPCIN has finished receiving the NC data, you can open the text file and check the data transferring results:

```
text.txt - Notepad
File  Edit  Format  View  Help

%_N_CYCLE999_MPF
;$PATH=/_N_CMA_DIR
PROC CYCLE999 (REAL HEIGHT, REAL WIDTH,  INT  RAD)     ;DATA  TRANSFER

N20  MSG (" cycle start")      ;text
N30  G1  G18  G91  G94  F500  S30  T1
N40  M17
→
```

See also

Siemens Website (http://www.ad.siemens.com.cn/download/)

6.3.5 Loading externally backed-up data

Restoring a start-up archive

1. In the "SYSTEM" operating area, press the "Archv." softkey.
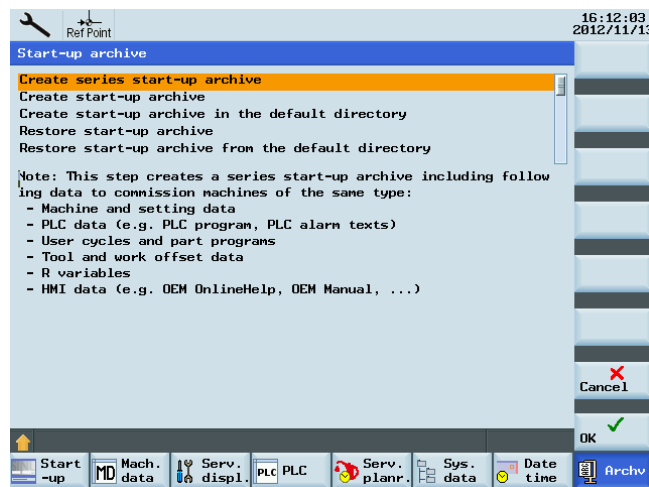
2. Select the option "Restore start-up archive" or "Restore start-up archive from the default directory", and press "OK". Note that the later option appears only after you have created an archive using the option "Create start-up archive in the default directory".
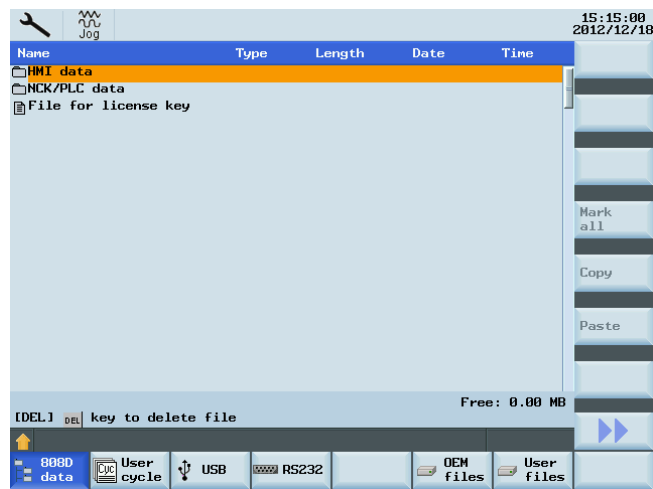
   If you use the option "Restore start-up archive", you also need to select the backup path and find the backed-up archive.
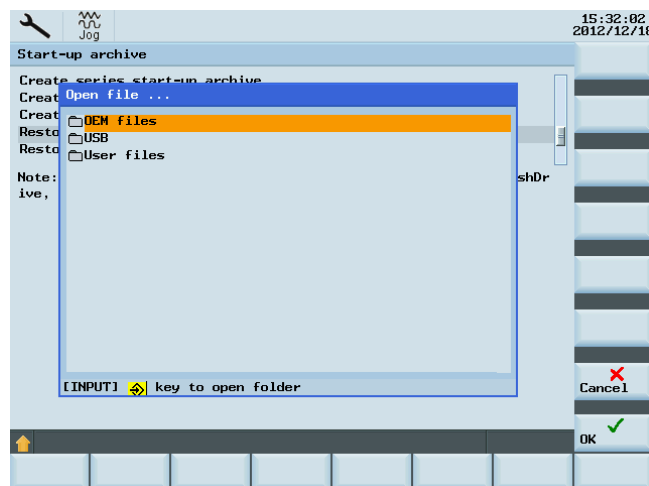
3. Press "OK" to confirm the archive information.

4. Press "OK" to continue and start restoring the start-up archive. The control system restarts to complete restoring the archive.

Loading separate files

1. In the "SYSTEM" area, press the "Sys. data" softkey.

2. Press a softkey (USB, RS232, OEM files, User files) according to the backup path of the file.

3. Find the file and press the "Copy" / "Receive" (for RS232 only) softkey.

4. Press the "808D data" softkey and paste the copied file to the folder "NCK/PLC data" or "HMI data" with the "Paste" softkey.

Updating software

7

You can update the control system using a USB stick connected via the USB interface at the front of the control system.

> **Note**
>
> You must back up the data of the control system (NC/PLC/HMI) before you start the update!
>
> For information about the data back-up, see section "Data backup (Page 303)".

Updating

To update the control system using a USB stick, proceed as follows:

1. Switch off the control system.

2. Plug the USB stick into the USB interface at the front of the control system.

3. Switch on the control system.

4. While the control system boots, once the message "Press the SELECT key to enter set-up menu" appears, press <SELECT>.

   Select the option of "Software update" in the Set-up menu, and press <INPUT>.

   Note: If you want to stop updating the control system at this point of time, you can switch off the control system and then remove the USB stick.

5. Select the update file and press <INPUT>.

6. The update process begins when the message "SW-update initiated" appears.

7. Wait until the message "Restoring complete. Switch off and remove data medium." appears.

8. Switch off the control system and remove the USB stick.

9. Switch on the control system.

10. The update process has been completed when the two error messages "004060 Standard machine data loaded" and "400006 Buffered PLC data deleted" appear.

11. Press <RESET> and then <ALARM CANCEL> to clear the alarms.

Appendix A

A

List of abbreviations

Abbreviation	Source of abbreviation	Meaning
ASCII	American Standard Code for Information Interchange	American coding standard for the exchange of information
AUTO		Operating mode "Automatic"
ASUP		Asynchronous subprogram
AUXFU	Auxiliary Function	Auxiliary function
BIN		Binary file
CNC	Computerized Numerical Control	Computerized numerical control
CPU	Central Processing Unit	Central processing unit
CRC	Cyclic Redundancy Check	Checksum test
CUTCOM	Cutter radius Compensation	Tool radius compensation
DB	Data Block	Data block in the PLC
DBB	Data Block Byte	Data block byte in the PLC
DBX	Data Block Bit	Data block bit in the PLC
DLL	Dynamic Link Library	Dynamic link library
DRAM	Dynamic Random Access Memory	Dynamic memory block
DRF	Differential Resolver Function	Differential resolver function (handwheel)
DRY		Dry run feedrate
I/O	Input/Output	Input/output
FIFO	First In - First Out	Method of storing and retrieving data in a memory
FRAME		Data set, coordinate conversion with the components zero (work) offset, rotation, scaling, mirroring
CRC	Cutter Radius Compensation	Cutter radius compensation
GUD	Global User Data	Global user data
HMI	Human Machine Interface	Controller user interface
HW	Hardware	Hardware
IBN		Commissioning
IK (GD)		Implicit communication (global data)
INC	Increment	Increment
IGBT	Insulated Gate Bipolar Transistor	Insulated gate bipolar transistor
IPO	Interpolator	Interpolator
ISO	International Standardization Organization	International Standardization Organization
JOG		"Jogging" operating mode
KV		Gain factor of control loop
K_v		Servo-gain factor

Abbreviation	Source of abbreviation	Meaning
LAD	Ladder Diagram	Ladder diagram
LED	Light Emitting Diode	Light emitting diode
LUD	Local User Data	Local user data
MCP	Machine Control Panel	Machine control panel
MD	Machine Data	Machine data
MDI	Manual Data Automatic	Manual input
MCS	Machine Coordinate System	Machine coordinate system
MPF	Main Program File	Main program (NC part program)
NC	Numerical Control	Numerical control
NCK	Numerical Control Kernel	Numerical control kernel
NCU	Numerical Control Unit	Hardware unit of the NCK
WO	Workpiece Offset	Zero offset
OB	Organization Block	Organization block in the PLC
OEM	Original Equipment Manufacturer	Original equipment manufacturer
OP	Operator Panel	Operator panel
OPI	Operator Panel Interface	Interface for connection to the operator panel
PC	Personal Computer	Personal computer
PCU	Programmable Control Unit	Programmable control unit
PCMCIA	Personal Computer Memory Card International Association	Standard for plug-in memory cards
PG	Programming device	Programming device
PLC	Programmable Logic Controller	Programmable logic controller
PPU	Panel Processing Unit	Panel-based control
QEC	Quadrant Error Compensation	Quadrant error compensation
RAM	Random Access Memory	Program memory that can be read and written to
REF POINT		"Reference-point approach" in JOG mode
REPOS		"Repositioning" in JOG mode
RPA	R parameter Active	Memory area on the NCK for R parameter numbers
SBL	Single Block	Single block
SBR	Subroutine	Subroutine (PLC)
SD	Setting Data	Setting data
SKP	Skip	Skip block
SPF	Subprogram File	Subprogram (NC)
SRAM	Static Random Access Memory	Static memory block
LEC	Leadscrew Error Compensation	Leadscrew error compensation
SW	Software	Software
TEA	Testing Data Active	Identifier for machine data
TO	Tool Offset	Tool offset
TOA	Tool Offset Active	Identifier (file type) for tool offsets
UFR	User Frame	User frame

Abbreviation	Source of abbreviation	Meaning
WAB		Smooth approach and retraction
Work		Workpiece coordinate system
T	Tool	Tool
TLC	Tool Length Compensation	Tool length compensation
TC	Tool Change	Tool change
TM	Tool Management	Tool management

Index