SIEMENS

SINUMERIK

SINUMERIK 808D, SINUMERIK 808D ADVANCED Diagnostics Manual

Diagnostics Manual

Preface

Fundamental safety instructions 1

Introduction 2

Operating in the system data operating area 3

Operating in the alarm operating area 4

SINUMERIK 808D/ SINUMERIK 808D ADVANCED alarms 5

System responses 6

SINAMICS V70 faults and alarms 7

SINAMICS V60 faults and alarms 8

Data backup 9

Updating software 10

Appendix A A

06/2017
6FC5398-6DP10-0BA4

Промэлектроника, sales@prom-elec.com

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.

Siemens AG
Division Digital Factory
Postfach 48 48
90026 NÜRNBERG
GERMANY

Document order number: 6FC5398-6DP10-0BA4
℗ 06/2017 Subject to change

Copyright © Siemens AG 2017.
All rights reserved

Preface

Applicable products

This manual is valid for the following control systems:

Control system	Software version
SINUMERIK 808D ADVANCED T (Turning) SINUMERIK 808D ADVANCED M (Milling)	V4.8.1: PPU16x.3 with spindle/feed servo system
SINUMERIK 808D (Turning) SINUMERIK 808D (Milling)	V4.8.1: PPU141.3 with feed servo system

Documentation components and target audience

End-user documentation	Target audience
Programming and Operating Manual (Turning)	Programmers and operators of turning machines
Programming and Operating Manual (Milling)	Programmers and operators of milling machines
Programming and Operating Manual (ISO Turning/Milling)	Programmers and operators of turning/milling machines
Programming and Operating Manual (Manual Machine Plus (MM+), Turning)	Programmers and operators of turning machines
Diagnostics Manual	Mechanical and electrical designers, commissioning engineers, machine operators, and service and maintenance personnel
Manufacturer/service documentation	**Target audience**
Commissioning Manual	Installation personnel, commissioning engineers, and service and maintenance personnel
Function Manual	Mechanical and electrical designers, technical professionals
Parameter Manual	Mechanical and electrical designers, technical professionals
Service Manual	Mechanical and electrical designers, technical professionals, commissioning engineers, and service and maintenance personnel
Readme file	
Third-party software - Licensing terms and copyright information	

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

Standard scope

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

Technical support

Country	Hotline [1]	Further service contact information:
Germany	+49 911 895 7222	• Worldwide Web site: https://support.industry.siemens.com/cs/ww/en/ • Chinese Web site: http://www.siemens.com.cn/808D
China	+86 400 810 4288	

[1] You can find more hotline information at the worldwide Web site given above.

EC Declaration of Conformity

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

Here, enter the number "**67385845**" as the search term or contact your local Siemens office.

Table of contents

Preface **3**

1 Fundamental safety instructions **7**

1.1 General safety instructions 7

1.2 Warranty and liability for application examples 7

1.3 Industrial security 8

2 Introduction **9**

2.1 Structure of the Diagnostics Manual 9

2.2 Alarm number ranges 10

2.3 Notes on the PLC interface signal address representation 11

3 Operating in the system data operating area **13**

3.1 Operating area overview 13

3.2 Setting the password 14

3.3 Setting startup function 17

3.4 Setting both system machine data and setting data 19

3.5 PLC diagnostics (manufacturer or Siemens access level) 24
3.5.1 Screen layout 25
3.5.2 Operating options 26
3.5.3 Displaying information on the program blocks 29
3.5.4 Displaying cross-references 30

3.6 Setting the HMI display 31
3.6.1 Setting the date and time 31
3.6.2 Adjusting the screen brightness 32
3.6.3 Configuring the operating area after startup 33

3.7 Managing the system data 33

3.8 Creating commissioning archives 36

3.9 Viewing the service info 37
3.9.1 Action log 38
3.9.2 Service MSG 39
3.9.3 Servo trace 41
3.9.4 Version/HMI details 43

3.10 Configuring the network functions 45
3.10.1 Establishing an Ethernet connection 45
3.10.2 Configuring the network drive 47
3.10.3 Configuring the firewall 48

3.11 Extending/deactivating the CNC lock function 49

3.12 Performing and monitoring maintenance tasks 51

4 Operating in the alarm operating area....53
5 SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms....55
5.1 System error alarms....55
5.2 NCK alarms....55
5.3 Cycle alarms....381
5.4 Drive alarms....537
5.5 PLC alarms....549
5.6 PLC user alarms....555
6 System responses....557
6.1 System reactions to SINUMERIK alarms....557
6.2 Cancel criteria for alarms....560
7 SINAMICS V70 faults and alarms....563
7.1 Overview....563
7.2 List of faults and alarms....565
8 SINAMICS V60 faults and alarms....579
8.1 Overview....579
8.2 Common faults and alarms....579
8.3 Errors during drive self-test....583
8.4 Other faults....583
9 Data backup....585
9.1 Internal data backup....585
9.2 External data backup....586
9.2.1 External data backup in a data archive....587
9.2.2 External data backup of system data....589
9.2.3 External data backup through the Ethernet interface....591
10 Updating software....595
A Appendix A....597

Fundamental safety instructions 1

1.1 General safety instructions

WARNING

Danger to life if the safety instructions and residual risks are not observed

If the safety instructions and residual risks in the associated hardware documentation are not observed, accidents involving severe injuries or death can occur.

- Observe the safety instructions given in the hardware documentation.
- Consider the residual risks for the risk evaluation.

WARNING

Malfunctions of the machine as a result of incorrect or changed parameter settings

As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.

- Protect the parameterization (parameter assignments) against unauthorized access.
- Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.

1.2 Warranty and liability for application examples

The application examples are not binding and do not claim to be complete regarding configuration, equipment or any eventuality which may arise. The application examples do not represent specific customer solutions, but are only intended to provide support for typical tasks. You are responsible for the proper operation of the described products. These application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.

1.3 Industrial security

Note

Industrial security

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Siemens products and solutions only represent one component of such a concept.

The customer is responsible for preventing unauthorized access to its plants, systems, machines and networks. Systems, machines and components should only be connected to the enterprise network or the internet if and to the extent necessary and with appropriate security measures (e.g. use of firewalls and network segmentation) in place.

Additionally, Siemens' guidance on appropriate security measures should be taken into account. For more information about industrial security, please visit:

Industrial security (http://www.siemens.com/industrialsecurity).

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends to apply product updates as soon as available and to always use the latest product versions. Use of product versions that are no longer supported, and failure to apply latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed at:

Industrial security (http://www.siemens.com/industrialsecurity).

⚠ WARNING

Unsafe operating states resulting from software manipulation

Software manipulations (e.g. viruses, trojans, malware or worms) can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.

Introduction 2

2.1 Structure of the Diagnostics Manual

Alarms

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

- NCK alarms (Page 55)
- Cycle alarms (Page 381)
- Drive alarms (Page 537)
- PLC alarms (Page 549)
- PLC user alarms (Page 555)
- SINAMICS V70 faults and alarms (Page 563)
- SINAMICS V60 faults and alarms (Page 579)

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

Structure of the NCK/cycle/drive/PLC alarm descriptions

The descriptions of the alarms have the following layout:

<Alarm No.> <Alarm text>

Parameters:

Explanation:

Reaction:

Remedy:

Program continuation:

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:

- Parameters
- Explanation
- Reaction (see Chapter "System reactions to SINUMERIK alarms (Page 557)")
- Remedy
- Program continuation (see Chapter "Cancel criteria for alarms (Page 560)")

Specification "%"

The specification "%" represents variables for an online parameter that is replaced on the control with a corresponding value. Since the SINUMERIK 808D/SINUMERIK 808D ADVANCED 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.

2.2 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 manual, only those number ranges are represented, which are valid for the specified product.

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 55)

Cycle alarms/messages

060 000 - 069 999	General cycle alarms

Cycle alarms (Page 381)

Drive alarms/messages

300 000 - 399 999	General drive alarms

Drive alarms (Page 537)

PLC alarms/messages

400 000 - 499 999	General PLC alarms

PLC alarms (Page 549)

2.3 Notes on the PLC interface signal address representation

Currently, PLC interface signal addresses are represented by the V structure on the HMI while the manual shows them by the DB structure.

See the following table for the relationship between the two representations.

V Structure		**DB Structure**	
Access	Example	Example	Access
Bit	V38000002.1	DB3800.DBX2.1	Bit
Byte	VB38000002	DB3800.DBB2	Byte
Word	VW38000002	DB3800.DBW2	Word
Double Word	VD38000004	DB3800.DBD4	Double word

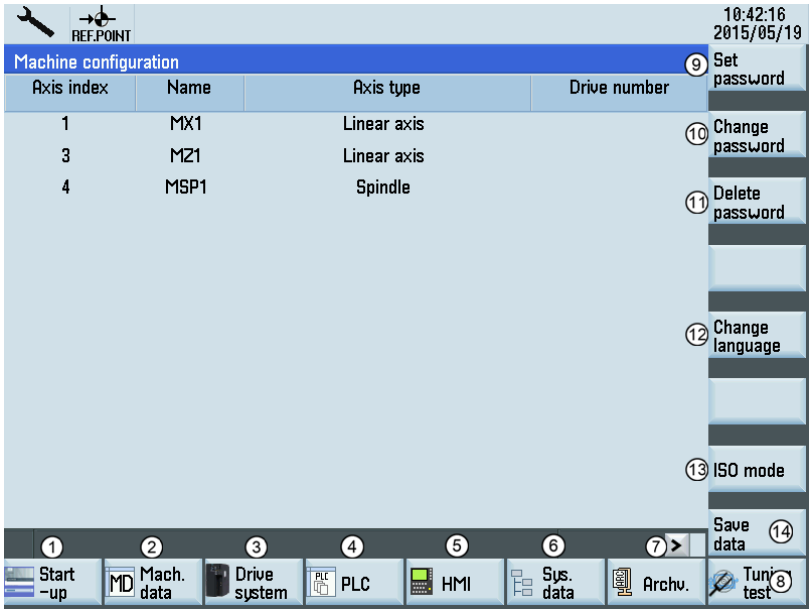
Operating in the system data operating area 3

3.1 Operating area overview

Softkey functions

+

Pressing this key combination allows you to enter the system data operating area. This operating area includes functions required for parameterizing and analyzing the NCK, the PLC, and the drive. 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 of the control system with PPU161.3 as an example.



① Sets the NC, PLC and HMI start up modes
② Sets the system machine data
③ Configures the connected drives and motors (PPU16x.3 only)
④ Provides PLC commissioning and diagnostics
⑤ Sets the system date and time and adjusts the brightness of the screen
⑥ Backs up and restores system data
⑦ Creates and restores startup archives, data archive
⑧ Performs the axis optimization (PPU16x.3 only)
⑨ Enters the corresponding password (manufacturer password, and end user password) for different access levels
⑩ Changes the password as per the corresponding access levels

⑪ Deletes the current password
⑫ Selects the user interface language. Note that the HMI is automatically restarted when a new language is selected.
⑬ Switches to the ISO programming mode
⑭ Saves the contents of the volatile memory into a non-volatile memory area

Note: Softkeys ③, ⑧, and ⑬ are visible only with the manufacturer password.

An extended horizontal softkey bar can be accessed via this key on the PPU. Two extended horizontal softkeys are provided:

Views the service information

Defines the maintenance planner

3.2 Setting the password

Overview

The control system provides a concept of protection levels for enabling data areas. Different protection levels control different access rights.

The control system delivered by Siemens is set by default to the lowest protection level 7 (without password). If the password is no longer known, the control system must be reinitialized with the default machine/drive data. All passwords are then reset to default passwords for this software release.

Note

Before you boot the control system with default machine/drive data, make sure that you have backed up your machine/drive data; otherwise, all data are lost after rebooting with default machine/drive data.

Protection level	Locked by	Area
0	Siemens password	Siemens, reserved
1	Manufacturer password	Machine manufacturers
2	Reserved	-
3-6	End user password (Default password: "CUSTOMER")	End users
7	No password	End users

Protection level 1

Protection level 1 requires a manufacturer password. With this password entry, you can perform the following operations:

- Entering or changing part of the machine data and drive data
- Conducting NC and drive commissioning

Protection level 3-6

Protection level 3-6 requires an end user password. With this password entry, you can perform the following operations:

- Entering or changing part of the machine data
- Editing programs
- Setting offset values
- Measuring tools

Protection level 7

If you have not set the password or protection level interface signal, the system sets the protection level to 7 automatically. With this password entry, you can perform the following operations:

- Editing programs
- Setting offset values
- Measuring tools

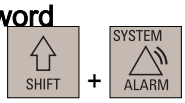
You can also set protection level 7 through the PLC user interfaces. For more information, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

In the menus listed below the input and modification of data depends on the set protection level:

- Tool offsets
- Work offsets
- Setting data
- Program creation/program correction

The number of machine data and drive data which can be read or modified depends on the protection level. You can set the protection level for these function areas with the display machine data (**USER_CLASS**...).

Setting the password



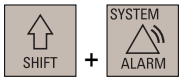
1. Select the system data operating area.
2. Open the following password setting window and then enter the desired password (default end-user password: CUSTOMER):

3. Confirm the input.

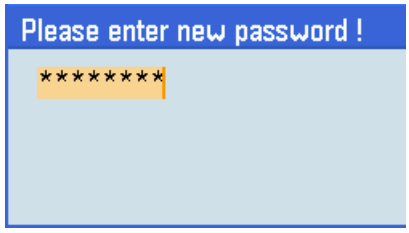
Changing/deleting the password

Note

To avoid unauthorized access to the controller, you must change the Siemens default passwords to your own ones.

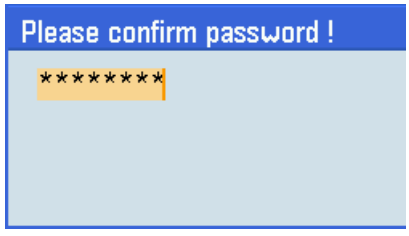


1. Select the system data operating area.
2. If you desire to change the existing password, press this softkey to open the following window and enter the new password:



If you desire to delete the existing password, proceed directly to Step 6.

3. Confirm the new password.
4. Enter the new password again in the following window:



5. Press this softkey to confirm the new password.

6. If you desire to delete the existing password, directly press this softkey.

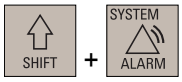
3.3 Setting startup function

Functionality

This softkey allows you to choose the NC, PLC, and drive startup modes. You can set up the startup modes by pressing the respective softkeys.

Selecting an NC startup mode

Proceed through the following steps to select an NC startup mode:

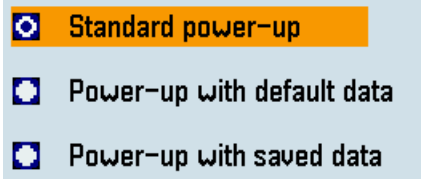


1. Select the system data operating area.

2. Press this softkey.

3. Press this softkey to open the window for selecting the NC startup mode.

4. Select a desired mode with the cursor keys. Three NC startup modes are available:



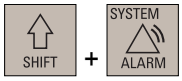
5. Press this softkey to confirm your selection. The system then restarts in the mode selected.

Note

You must stop the machine before selecting an NC startup mode.

Selecting a PLC startup mode

Proceed through the following steps to select a PLC startup mode:



1. Select the system data operating area.

2. Press this softkey.

3. Press this softkey to open the window for selecting the PLC startup mode.

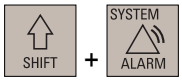
4. Select a desired mode with the cursor keys. Two PLC startup modes are available:

5. Use this key to select/deselect the PLC debugging.

6. Press this softkey to confirm your selection. The system then restarts in the mode selected.

Selecting a drive startup mode (PPU16x.3 only)

Proceed through the following steps to select an drive startup mode:

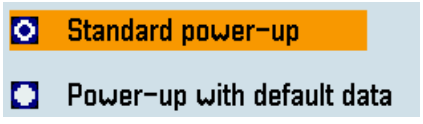


1. Select the system data operating area.

2. Press this softkey.

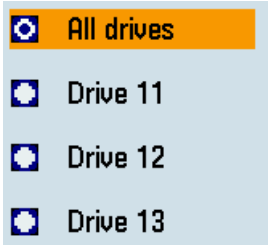
3. Press this softkey to open the window for selecting the drive startup mode.

4. Select a desired mode with the cursor keys. Two drive startup modes are available:



5. Press this softkey to confirm your selection, and then the drive list window opens displaying all the connected drives.

6. Select all drives or one specific drive using the cursor keys.

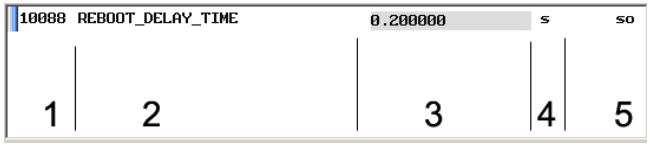


7. Press this softkey to confirm your selection. The selected drive(s) will restart in the mode selected.

3.4 Setting both system machine data and setting data

Machine data structure

Any changes in the machine data have a substantial influence on the machine. The following describes the machine data structure in detail:



No.	Meaning
1	MD number
2	Name
3	Value
4	Unit
5	Effective im: immediately cf: with confirmation re: after reset po: after power on

CAUTION

Machine damage

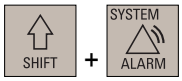
Incorrect parameterization may result in destruction of the machine.

Setting basic machine data

The control system provides easy-to-use data lists for beginner users. You can use the basic data list for quick access of common data.

Setting basic NC data

In the basic NC data list, the general, axis, and channel MD are integrated in one screen. Proceed through the following steps to set the basic NC data:



1. Select the system data operating area.

2. Press this softkey, and the window of basic NC data opens by default.

3. Locate the machine data which you desire to set.

You can also search for an MD using the following softkeys:

Searches for the desired number or the name (or a part of the name) of the machine data

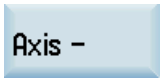
Continues searching for the next match

4. Enter the desired value in the blank and press this key to confirm.

5. Press this softkey to activate the modified value(s) after you finish setting the machine data.

Further softkey operations are available for setting the basic NC data:

Switches to the next axis



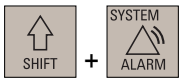
Switches to the previous axis

Resets the value of the selected MD to default

Undoes the modified values and recalls the lastly used values

Setting basic drive data (PPU16x.3 only)

Several common drive parameters are provided in the basic drive data list. Proceed through the following steps to set the basic drive data:



1. Select the system data operating area.

2. Enter the window of basic drive data through the following softkey operations:

3. Locate the parameter which you desire to set.

   You can also search for a parameter using the following softkeys:

   Search
   Searches for the desired number or the name (or a part of the name) of the parameter

   Continue search
   Continues searching for the next match

4. Enter the desired value in the blank and press this key to confirm.

5. Press this softkey to save the modified value(s) after you finish the parameter setting.

   Save paramet.

   Further softkey operations are available for setting the basic drive data:

   Drive +
   Switches to the next drive

   Drive –
   Switches to the previous drive

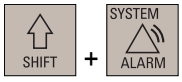
   Default
   Resets the value of the selected parameter to default

   Undo
   Undoes the modified values and recalls the lastly used values

Setting both machine data and setting data in expert list

Five groups of machine data and three groups of setting data are included in the expert list.

Setting the machine data



1. Select the system data operating area.

2. Open the expert data list through the following softkey operations:

   MD Mach. data → **Expert list**

   The following five groups of machine data are available:

   General MD
   General machine data

   Channel MD
   Channel-specific machine data

   Axis MD
   Axis-specific machine data

Drive machine data. This softkey is available on the PPU16x.3 only.

Display machine data. This softkey is available on the extended horizontal softkey bar which can be accessed via the following key:

3. Locate the machine data which you desire to set.

You can also search for an MD using the following softkeys:

Searches for the desired number or the name (or a part of the name) of the machine data

Continues searching for the next match

4. Enter the desired value in the blank and press this key to confirm.

5. Press this softkey to activate the modified value(s) after you finish setting the machine data.

Further softkey operations are available for setting the machine data:

Resets the value of the selected MD to default

Undoes the modified values and recalls the lastly used values

Provides various display filters for the active machine data group

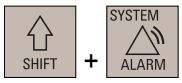
For the axis-specific machine data, two additional softkeys are also provided:

Switches among available axes/spindle

While for the drive machine data, the following two softkeys are additionally provided:

Switches among available drives

Setting the setting data



1. Select the system data operating area.

2. Open the expert data list through the following softkey operations:

3. Press this key to view the extended softkeys. The following three groups of setting data are available:

General setting data

Channel-specific setting data

Axis-specific setting data

4. Locate the setting data which you desire to set.

You can also search for an SD using the following softkeys:

Searches for the desired number or the name (or a part of the name) of the setting data

Continues searching for the next match

5. Enter the desired value in the blank and press this key to confirm.

6. Press this softkey to activate the modified value(s) after you finish setting the data. Further softkey operations are available for the data setting:

Resets the value of the selected setting data to default

Undoes the modified values and recalls the lastly used values

Provides various display filters for the active setting data group

For the axis-specific setting data, two additional softkeys are also provided:

Switches among available axes/spindle

References

You can find a description of the machine data and setting data in the following manufacturers' documents:

SINUMERIK 808D/SINUMERIK 808D ADVANCED Parameter Manual

SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual

3.5 PLC diagnostics (manufacturer or Siemens access level)

This function is accessible with the manufacturer or Siemens access level.

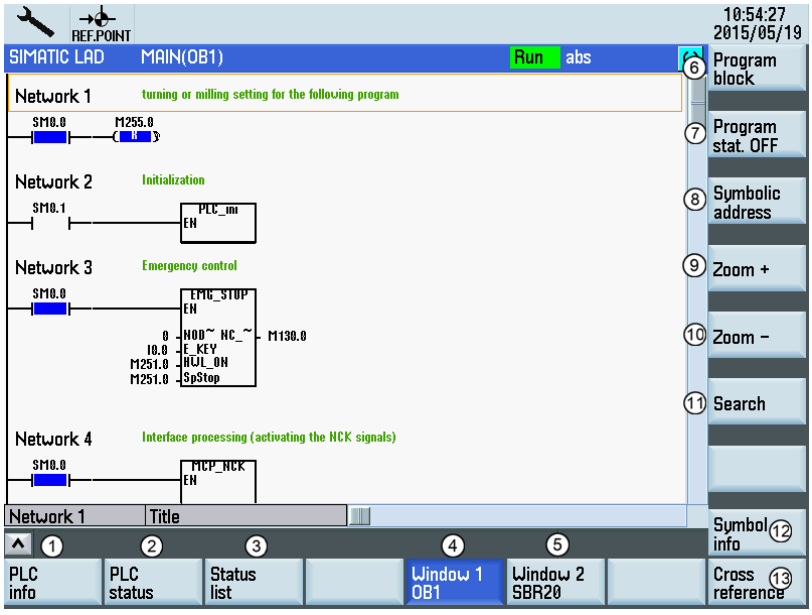
Functionality

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 data operating area.

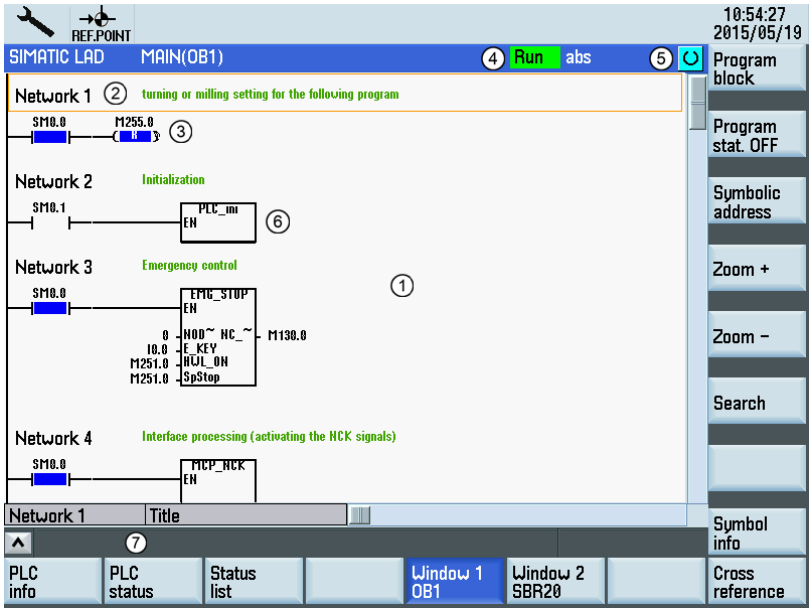
Softkey functions

You can open the project stored in the permanent memory through the following operations:



① Displays PLC properties
② Monitors and changes the values of the operands during program execution
③ Displays and modifies PLC signals
④ Displays any logical and graphical information of a program block
⑤ Displays any logical and graphical information of a program block in another window
⑥ Displays the list of cross references
⑦ Displays the logic and graphic information of the selected program block
⑧ Activates or deactivates the program status display
⑨ Switches between the absolute and symbolic representation of the operands. The softkey labelling changes accordingly.
⑩ Zooms in the application area
⑪ Zooms out the application area
⑫ Searches for operands
⑬ Displays all symbolic identifiers used in the selected network

3.5.1 Screen layout



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 is stopped
- **Status of the application area**
  - Sym: Symbolic representation
  - abs: Absolute representation

⑤ **Display of the active keys, e.g.**

⑥ **Focus**

Performs the tasks of the cursor

⑦ **Tip line**

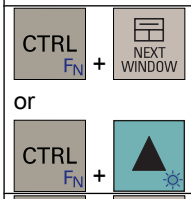
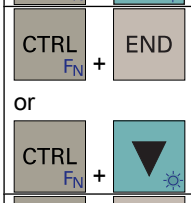
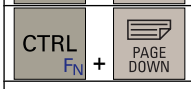
contains notes for searching

3.5.2 Operating options

Key operations

In addition to the softkeys and the navigation keys, this area provides still further key combinations. The cursor keys move the focus over the PLC user program. When reaching the window borders, it is scrolled automatically.

Key combination	Action
NEXT WINDOW or CTRL FN + ◀	To the first line of the row
END or CTRL FN + ▶	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

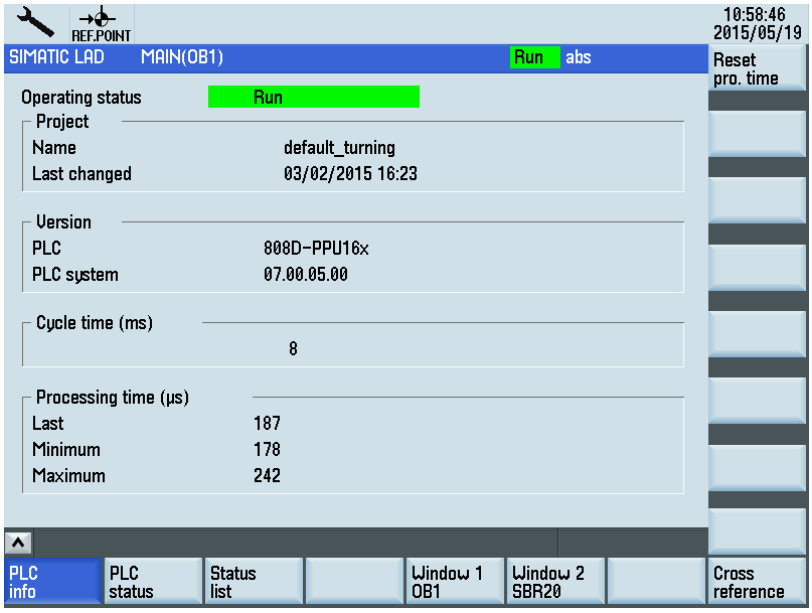
Key combination	Action
	Up a field
	Down a field
	To the first field of the subroutine
	To the last field of the subroutine
	Opens the previous program block in the same window
	Opens the next program block in the same window
	The function of the Select key depends on the position of the input focus. • Table line: Displays the complete text line • Network title: Displays the network comment • Command: Displays the complete operands
	If the input focus is positioned on a command, all operands including the comments are displayed.

Displaying the PLC info

Press this softkey on the PLC program start screen 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



The processing time data is reset by pressing this softkey.

Searching for operands

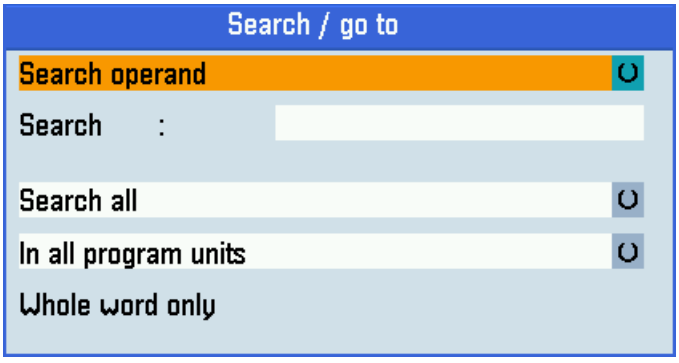
In big programs, you can use the search function to quickly reach the desired positions.

To search for operands, follow these steps:

1. Use this softkey to switch between the absolute and symbolic representation of the operands. After you press this softkey, it switches to the following:

2. Press this softkey to open the search dialog.

3. Use this key on the PPU to select the desired search options, and enter the search term in the input field.



4. Press this softkey to start the search, or press the following softkey to exit:

3.5.3 Displaying information on the program blocks

Functionality

You can display any logical and graphical information of a program block in the program windows. 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

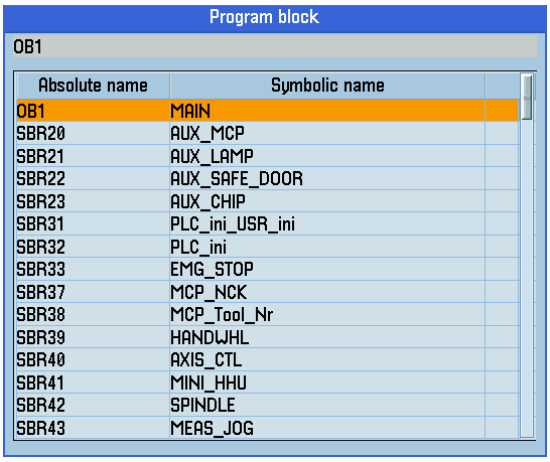
Further information

- Properties
  Symbolic name, author and comments
- Local variables
  Name of the variable, variable type, data type and comment

Operating sequence

Program block

1. Press this softkey to open the program block window.



Protected

2. If a program block is protected by a password, press this softkey 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 PLC Programming Tool.

Open

3. Select the desired program block and press this softkey to open it.

4. After the selected program block is opened, you can press the following softkeys to switch the display information.

Symbolic address / **Absolute address**

Further softkeys are available in the program block window as follows:

Proper-ties Displays additional information of the selected program block

Local variables Displays the table of local variables of the selected program block

3.5.4 Displaying cross-references

Cross reference

Use this 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.

Operating sequence

To display the cross references, follow these steps:

Cross reference

1. Press this softkey in the PLC program window. The list of cross references appears and the operands are displayed according to the absolute address by default.

Symbolic address

2. Press this softkey to switch between the absolute and symbolic representation of the components.

3. You can open the appropriate program segment directly in the window using the following softkeys:

   Open in window 1 | Open in window 2

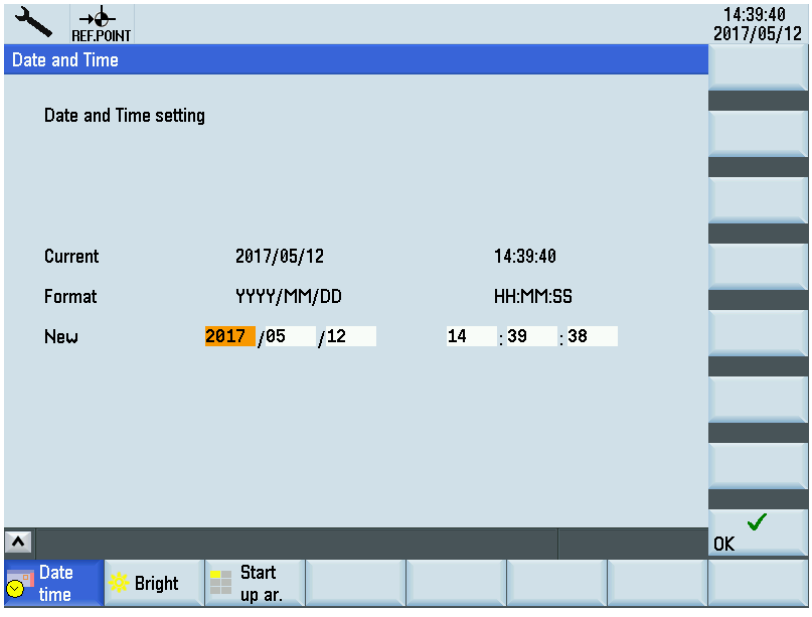
Search

4. You can also search for an operand in the cross reference list using this softkey.

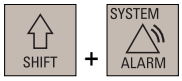
3.6 Setting the HMI display

3.6.1 Setting the date and time

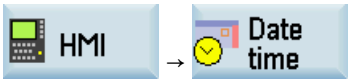
At delivery, the system date and time remain at the factory settings, and thus you must manually modify the date and time in the window as follows.



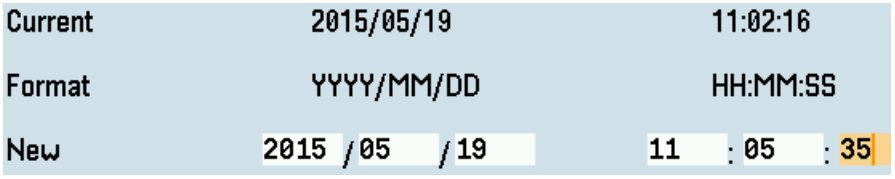
Operating sequence



1. Select the system data operating area.
2. Open the date and time setting window through the following softkey operations:



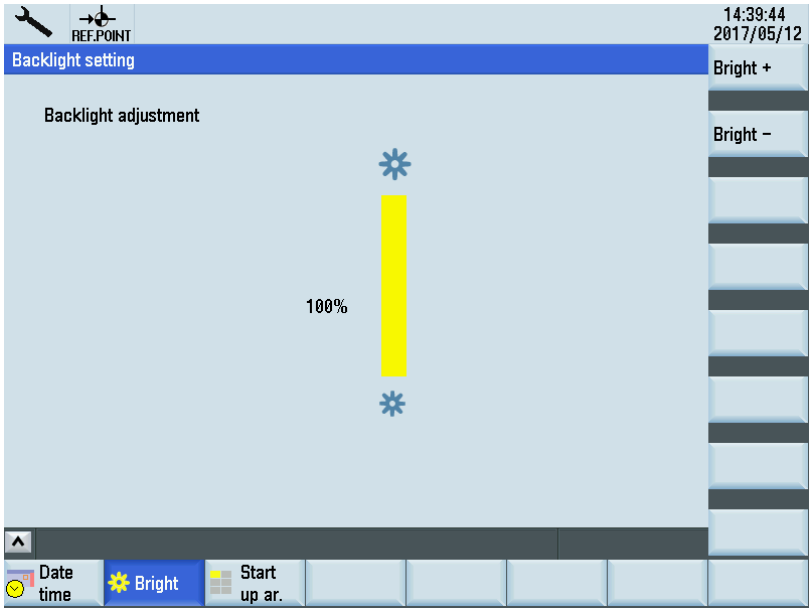
3. Enter the date and time in the specified format.



4. Press this softkey to confirm your settings.

3.6.2 Adjusting the screen brightness

You can open the window for adjusting the brightness of the HMI screen through the following operations:



The following softkey operations are available:

Bright +	Pressing this softkey increases the brightness of the HMI screen.
Bright −	Pressing this softkey decreases the brightness of the HMI screen.

3.6.3 Configuring the operating area after startup

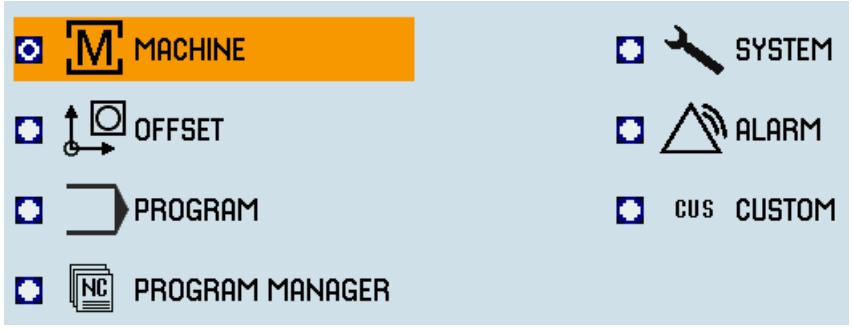
The machining operating area is displayed by default after the startup of the control system. Alternatively, you can select another operating area which you desire to enter after the system starts up.

Operating sequence

SHIFT + SYSTEM ALARM

1. Select the system data operating area.
2. Open the window for setting the startup operating area through the following softkey operations:

   HMI → Start up ar.
3. Use the cursor keys to select the desired operating area in the following window:



Change start ar.

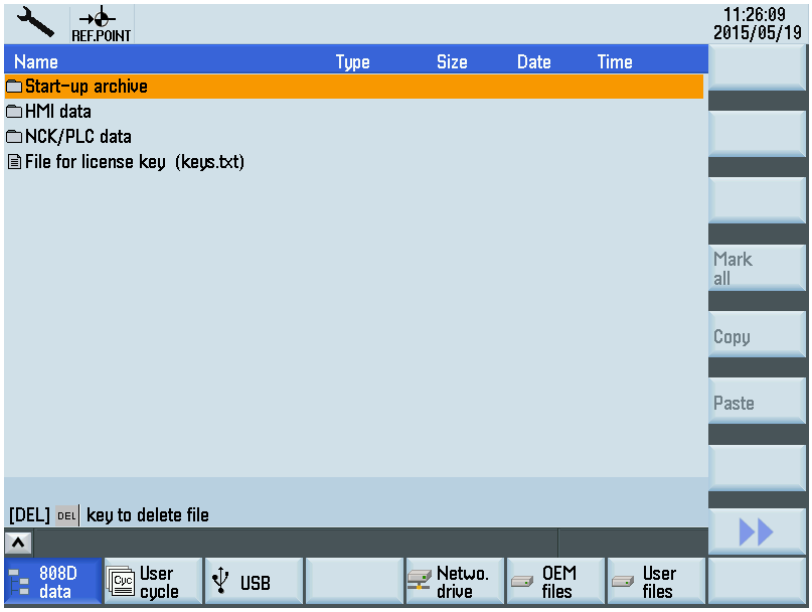
4. Press this softkey to confirm your setting, and the configured operating system will display after the control system restarts.

3.7 Managing the system data

Overview

Sys. data

By pressing this softkey in the system data operating area, you can enter the window as follows:



Three folders and one file are available in this window. Note that the last three entries are visible with the manufacturer password only. You can import/export the subfolders or single files for backup or other customized purposes.

Managing the start-up archives

Start-up archive

INPUT

Select this folder and press this key to enter. This folder is used to store the startup archives for the prototype machine or for the series commissioning. For more information about creating a startup archive, refer to Section "Creating commissioning archives (Page 36)".

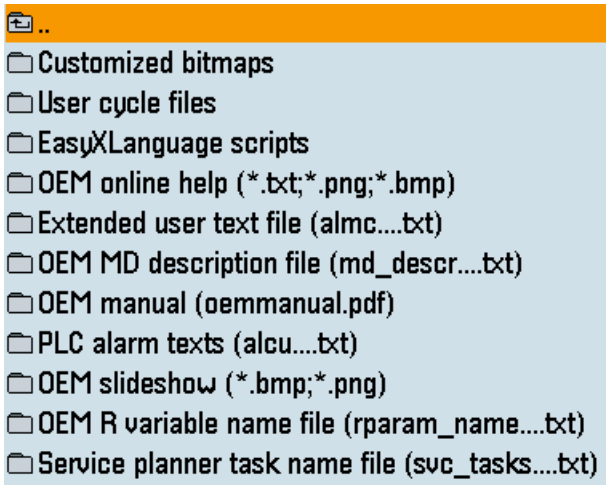
You can copy and paste the subfolders or single files in this folder for backup purposes.

Managing the HMI data

HMI data

INPUT

Select this folder and press this key to enter. There are 11 subfolders available in this folder as follows:

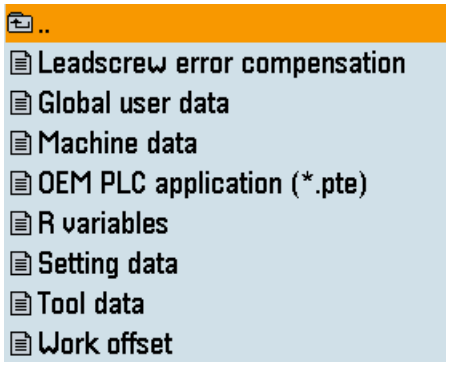


You can import/export the subfolders or single files in this window for backup or other customized purposes.

For more information, refer to the SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual and Commissioning Manual.

Managing the NCK/PLC data

Select this folder and press this key to enter. There are eight files available in this folder as follows:



You can copy and paste these files for backup purpose.

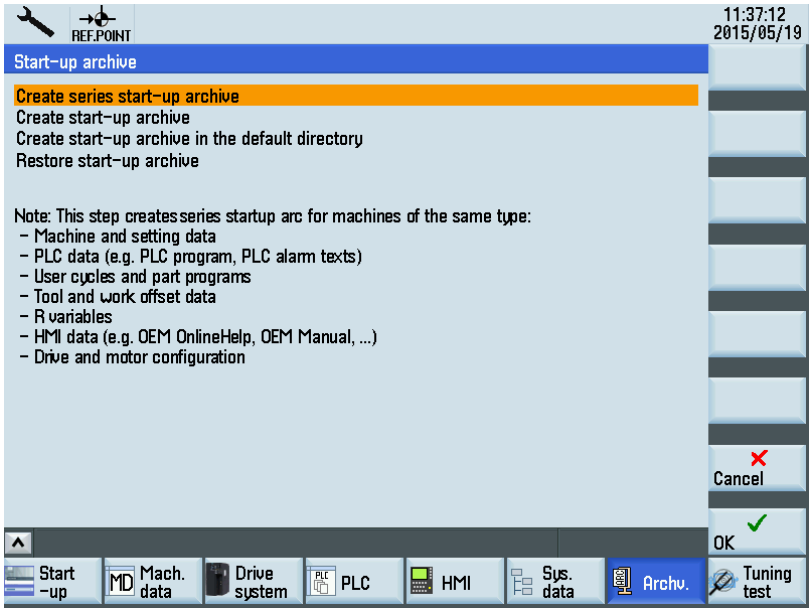
Managing the license key

This file contains the license key information of the control system. You can import/export it for backup purpose.

3.8 Creating commissioning archives

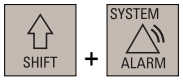
Functionality

Pressing this softkey allows you to create or restore a data archive in the following window.



Note that some entries in this window are visible with the manufacturer password only. For more information, see section "External data backup in a data archive (Page 587)".

Operating sequence



1. Select the system data operating area.

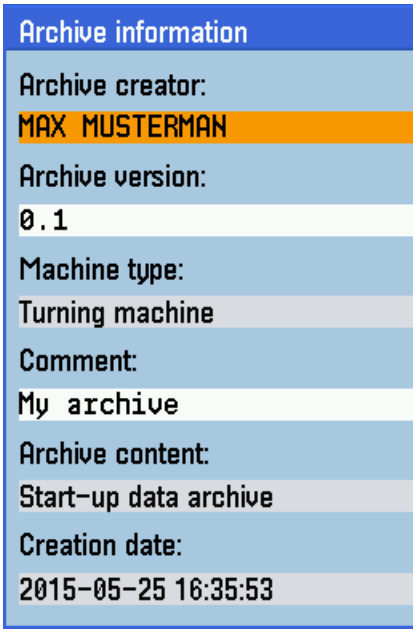
2. Press this horizontal softkey to open the startup archive window.

3. Use the cursor keys to select an option to create a startup archive.

4. Press this softkey to confirm, and the file saving dialog appears.

5. Select the desired storage folder and press this key to enter it.

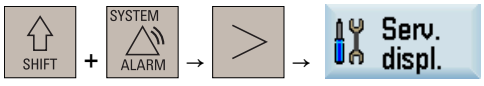
6. Press this softkey to confirm and the archive information dialog opens, for example:

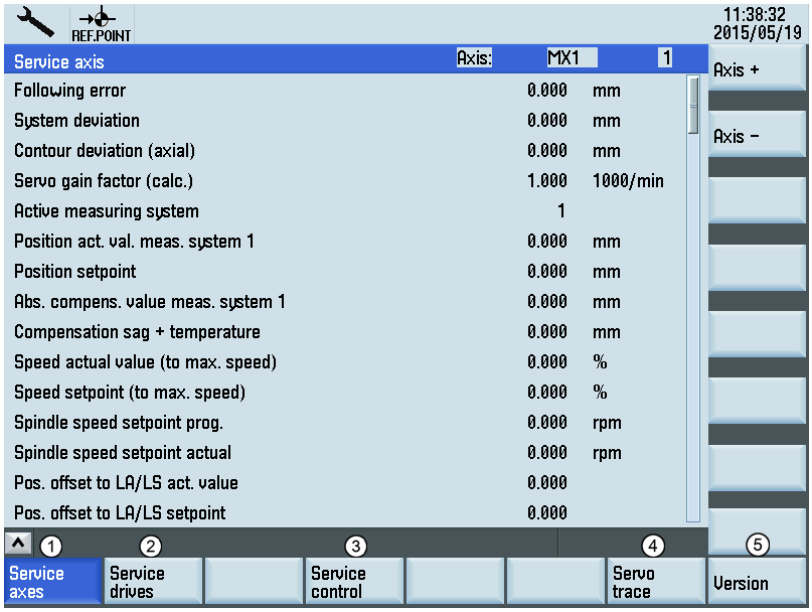


7. Specify the properties of the archive and press this softkey to start creating the archive file in the selected folder.

3.9 Viewing the service info

You can view the service information through the following operations:





① Displays information of the axes.

② Displays information of the drives. Note that this softkey is available on the PPU16x.3 only.

③ Displays the action log for service events, provides the possibility of sending service messages, and configures the Ethernet connection.

④ Compares the defined machine data with the actual data, and adjust the parameters if necessary. For more information, see Section "Servo trace (Page 41)".

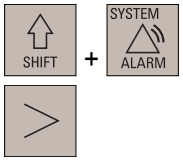
⑤ Displays the version numbers and the date of creation of the individual CNC components. For more information, see Section "Version/HMI details (Page 43)".

3.9.1 Action log

Overview

The action log function is provided for recording the service events on the PPU. You can view or save the action log as desired.

Viewing the action log



1. Select the system data operating area.
2. Press this key to view the extended softkeys.
3. Open the action log window through the following softkey operations:

4. The window opens displaying the action log for the service events.

5. You can press this softkey to set the filter conditions for the data to be displayed.

You can choose to display all data or certain data groups with this key.

6. To search for a specific event, press this softkey.

7. You can save the action log to the CF card or a USB stick (if inserted) using this softkey.

Note

Alternative method to save the action log

Pressing <CTRL + S> when you are in any operating area (on PPU16x.3 only) saves the action log to the connected USB stick. In addition, it automatically creates a startup archive on the USB stick.

3.9.2 Service MSG

Overview

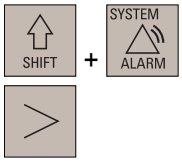
The service message function provides the possibility of sending message texts/messages to a file to be saved internally on the control system or externally on an USB stick.

Message texts/messages include:

- Programmed messages from part program
- Alarms

Settings for the message sending

Proceed through the following steps to configure the settings for the message sending:



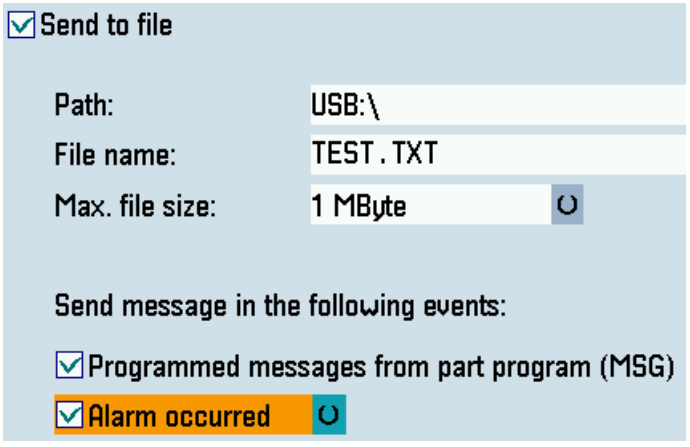
1. Select the system data operating area.

2. Press this key to view the extended softkeys.

3. Open the service message window through the following softkey operations:

4 Press this softkey to open the file setup window.

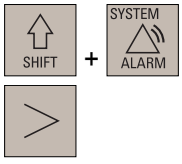
5 Use this key to select or deselect the settings as desired and enter the file name, for example:



6. Press this softkey to save the settings and return to the service message main screen.

Viewing the error log

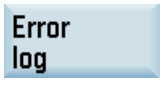
All messages with the associated error information, where an error occurred when processing them, are saved in the error log. To view the error log, proceed through the following steps:



1. Select the system data operating area.

2. Press this key to view the extended softkeys.

3. Open the service message window through the following softkey operations:



4. Press this softkey to open the error log window. The error log can be deleted using the following softkey:

3.9.3 Servo trace

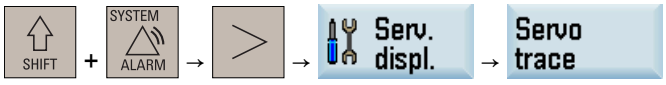
Overview

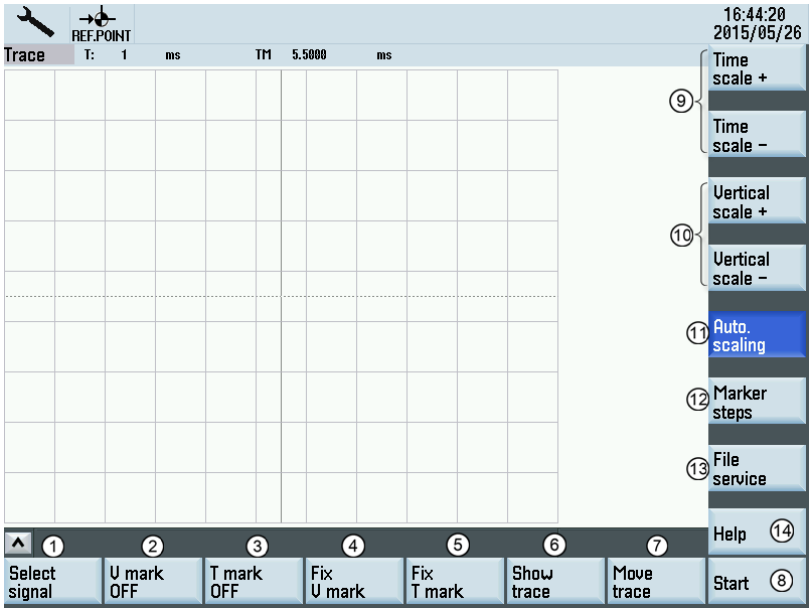
An oscilloscope function is provided for the purpose of optimizing the drives. This enables the following graphical representations:

- velocity setpoint
- contour violation
- following error
- actual position value
- position setpoint
- exact stop coarse/fine

Viewing the servo trace

You can open the servo trace main screen through the following operations:





① Parameterizes the measuring channel
② Hides/shows the vertical gridlines
③ Hides/shows the time gridlines
④ Determines the differences in the vertical direction
⑧ Starts or stops a servo trace
⑨ Zooms in/out the time basis
⑩ Increases/reduces the resolution (amplitude)
⑪ Shows the vertical or time grindlines automatically

⑤ Determines the differences in the horizontal direction

⑥ Hides/displays the selected diagrams in various traces

⑦ Moves the trace

⑫ Defines the step sizes of the markers

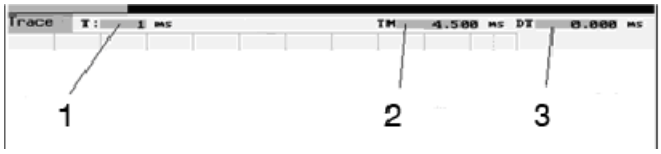
⑬ Saves or loads the trace data

⑭ Provides the help info

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

Selecting signal

To parameterize the measuring channel, follow these steps:

1. Press this softkey on the servo trace main screen to open the window for signal selection.

2. Use this key to choose the axis, signal type and enable tracing in the respective traces.

3. Set the parameters for the measuring time and the trigger type for the traces. The maximum measuring time is 6133 ms.

4. Press this softkey to confirm the settings.

To abort the settings and return, press this softkey.

Defining the marker steps

To define the marker steps, follow these steps:

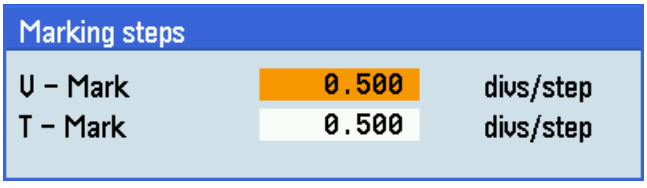
Marker steps

1. Press this softkey on the servo trace main screen to open the marker increment window.

2. Select the desired increment value from the vertical softkeys "V=0.1/0.5/1.0" or "T=0.1/0.5/1.0".

   Other

   For other step sizes, press this softkey and specify in the following window as desired:



   Back

3. Press this softkey to return after you set the desired marker steps.

   SHIFT

   If the trace exceeds the current screen, press this key and 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:

File service

1. Press this softkey on the servo trace main screen to open the data saving dialog.

2. Type the desired file name without extension in the input field.

   Save

3. Use this softkey to save the data with the specified name in the part program directory. The file then can be exported, and the data can be processed in Microsoft Excel.

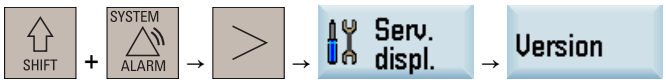
   Load

   You can also use this softkey to load the specified file and graphically display the data.

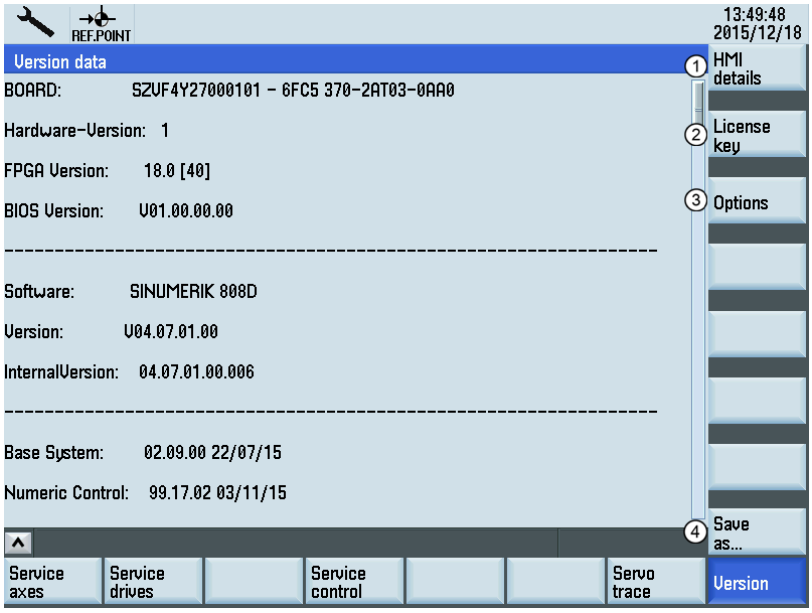
3.9.4 Version/HMI details

Overview

You can open the version data main screen through the following operations:



This window displays the version numbers and the date of creation of the individual CNC components.



① Displays the operator programs with the version numbers

② Opens the license configuring window to perform the following operations:

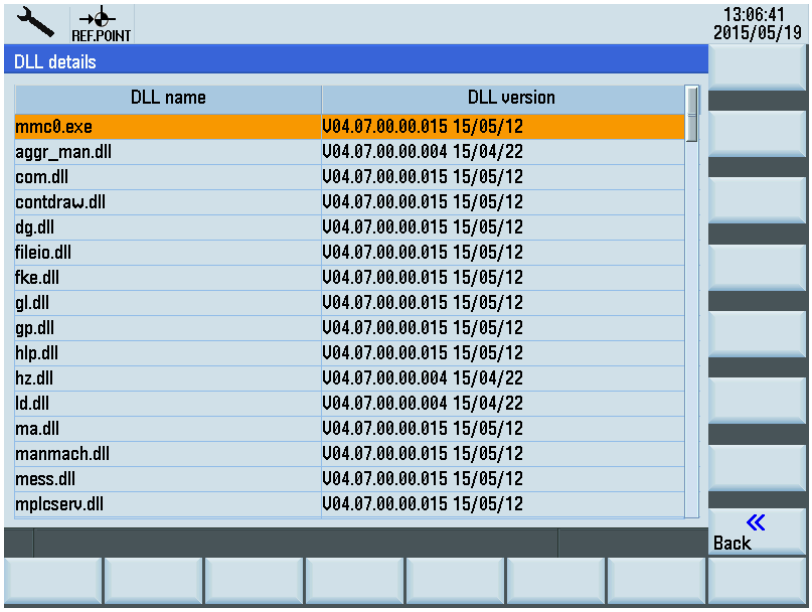
- Entering the license key
- Importing the "*.clc" file for activating, extending, or deactivating the CNC lock function. For more information, see Section "Extending/deactivating the CNC lock function (Page 49)".

③ Activates the licensed optional functions. This softkey is accessible with the manufacturer or higher-level password only.

④ Saves the version data as a file

HMI details

Pressing this softkey allows you to open the following window:



This window is intended for servicing. All programs provided by the HMI are displayed with their version numbers. By reloading software components, the version numbers can be different from each other.

3.10 Configuring the network functions

3.10.1 Establishing an Ethernet connection

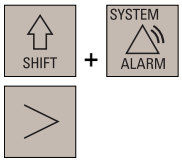
Ethernet connections

You can establish a network connection between the control system and a computer via the Ethernet interface X130. The following Ethernet connections are possible:

- Direct connection: connecting the control system directly to a computer
- Network connection: integrating the control system into an existing Ethernet network

Establishing a direct connection

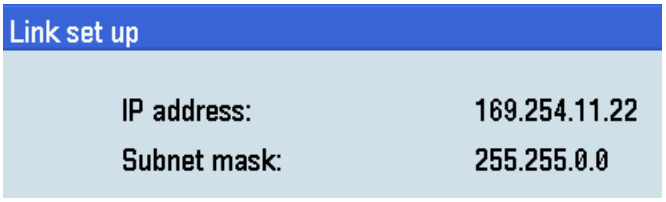
Proceed as follows to establish a direct connection between the control system and a computer:



1. Connect the control system with the computer using an Ethernet cable.
2. Select the desired operating area on the PPU.
3. Press this key to view the extended softkeys.

4. Set up a direct connection on the control system through the following softkey operations:

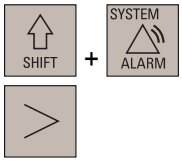
   The following dialog pops up on the screen:

Establishing a network connection

Proceed as follows to establish a network connection:

1. Connect the control system with the local network using an Ethernet cable.



2. Select the desired operating area on the PPU.

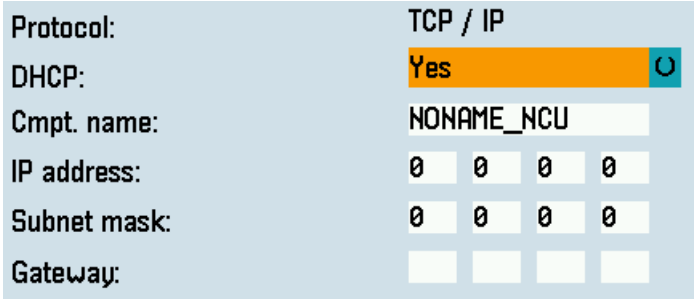
3. Press this key to view the extended softkeys.

4. Enter the main screen of the service control options through the following softkey operations:

5. Press this softkey to enter the window for the network configuration.

   Note: make sure this vertical softkey is not selected.

6. Configure the network as required in the following window:

You can configure the DHCP with this key.

Note: if you select "No" for DHCP, you must enter the IP address (which must belong to the same network as that of your PC) and subnet mask manually.

7. Press this softkey to save the configuration. If you select "Yes" for DHCP, you also need to restart the control system to activate the network configuration.

3.10.2 Configuring the network drive

A connected network drive allows you to access a shared directory on your computer from the control system. The network drive functions based on the Ethernet connection between the control system and a computer. For more information about how to establish an Ethernet connection, see Section "Establishing an Ethernet connection (Page 45)".

Creating and connecting a network drive

Proceed as follows to create and connect a network drive:

1. Share a directory on your local disk on your computer.

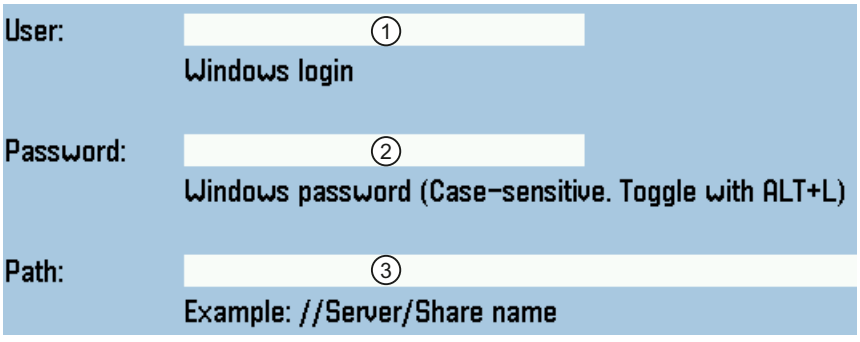
2. Select the program management operating area.

3. Press this softkey to go to the network drive directory.

4. Press this softkey to go to the window for configuring the network drives.

5. Press this key to select a drive identifier: N1, N2, or N3.

6. Move the cursor to the following input fields:



①: Enter the user name of your Windows account

②: Enter the logon password (case sensitive) of your Windows account

③: Enter the IP address of the server and the share name of the shared directory on your computer. Example: //140.231.196.90/808D

OK

7. Press this softkey to confirm and the configured network drive appears on the screen as follows. The drive icon is yellow if the network drive is connected successfully; otherwise, the icon is gray.

N1://140.231.196.90/808D

Delete net.drive

You can delete a selected network drive using this softkey.

Note

After you properly configure all the settings for the direct connection between the control system and the network drive, if the network drive connection is still invalid, contact your Windows system administrator for possible problems with your operating system configuration.

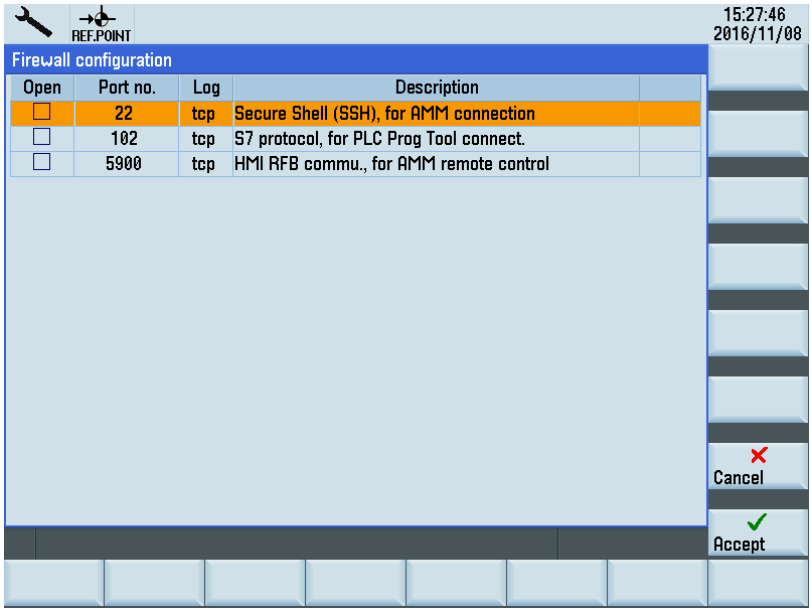
3.10.3 Configuring the firewall

Configuring the firewall

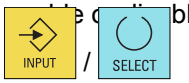
Secure access and communication is achieved through the security function of the integrated firewall. You can open the window for firewall configuration through the following operations:

SHIFT + SYSTEM ALARM → > → Serv. displ. → Service control → Service firewall

Configurable ports are listed in the following window:



The ports are disabled by default and can be enabled when necessary. To change the port status, select the relevant port using the cursor keys and press either of the following keys to ble the port:



WARNING

Death, severe personal injury and/or property damage due to improper firewall configuration

Improper firewall configuration may cause network security risks, for example, data leakage, virus invasion, and hacker attack. This may lead to incorrect parameterization or machine malfunction, which in turn can result in death, severe injuries and/or property damage.

- Do not use the control system inside a network infrastructure without an additional security product.
- You must make sure you disable all ports not needed in the firewall configuration.

Note

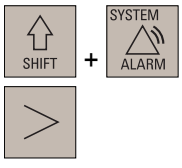
After you disable a communication port, the existing connection established earlier via this port will not be disabled until you manually disconnect it or the control system restarts/powers off. Therefore, it is recommended that you disconnect the established connections or restart the control system after disabling the ports.

3.11 Extending/deactivating the CNC lock function

To extend/deactivate the CNC lock function, you must import the corresponding activation/ deactivation file (.clc format) provided by the machine manufacturer into the control system. The file can be imported either directly via an Ethernet connection (network drive) or alternatively via a storage medium (for example, USB memory stick). The control system must be in the reset state for the import.

Importing the activation/deactivation file

Proceed through the following steps to import the file:



1.
   - To import the file via USB, store the file in a USB memory stick and insert the USB memory stick into the USB interface at the front of the PPU.
   - To import the file via Ethernet, store the file in a shared folder (network drive) on your computer and connect the network drive via Ethernet connection (Page 47).
2. Select the system data operating area.
3. Press this key to view the extended softkeys.
4. Open the license key dialog box through the following softkey opera-

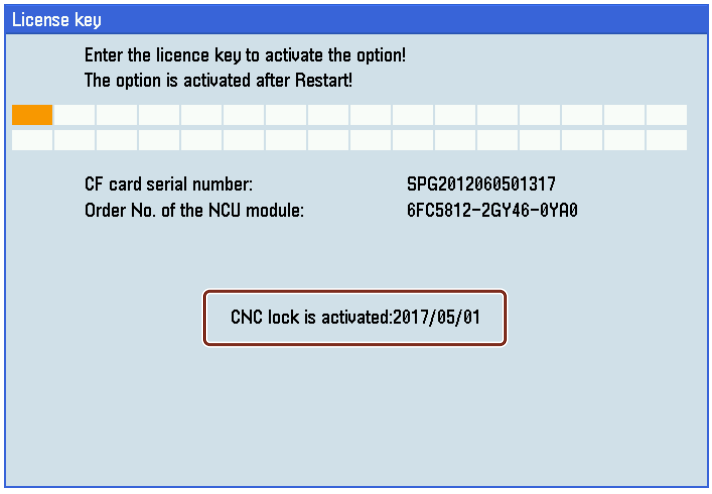
5. Press this softkey to open the file opening dialog box.

6. Select the target directory (USB or network drive) and press this key to open it.

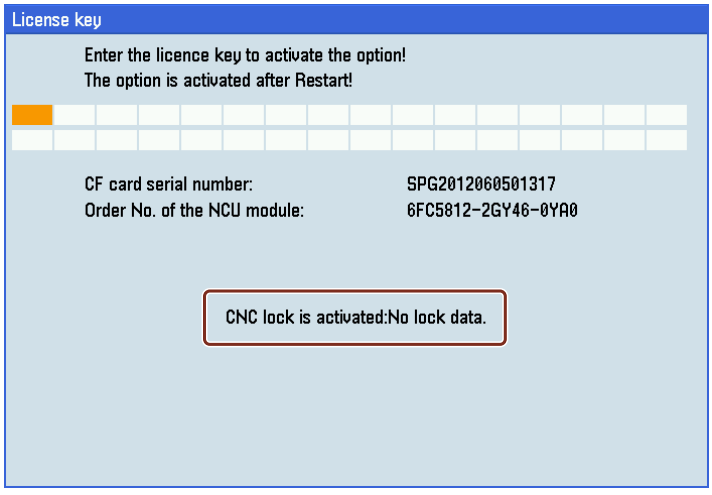
7. Locate the desired activation/deactivation file and press this softkey to import.

   After the file is imported successfully, you can check the CNC lock status on the HMI screen, for example:

   - CNC lock function extended (with a new lock date):



   - CNC lock function deactivated (with no lock date):



Note

If an error occurs when importing the activation file, an error-specific alarm will be issued. The state of the CNC lock function remains unchanged.

Note

We recommend that you create a complete commissioning archive over all control system components after deactivating the CNC lock function. If necessary, this commissioning archive can be used to recommission the control system without re-deactivating the CNC lock function.

Faulty settings of date

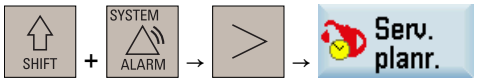
If a date earlier than the actual date is set for activated CNC lock function, alarm 8065 is issued after NC restart and then NC start is disabled. In this case, you must correct the date and perform an NC restart again to clear the alarm.

If during the correcting, a future date is set inadvertently, alarm 8066 is issued. Provided no NC restart has been performed, the date can still be corrected. After NC restart, a date set in the future is considered as being an actual date and can no longer be reset.

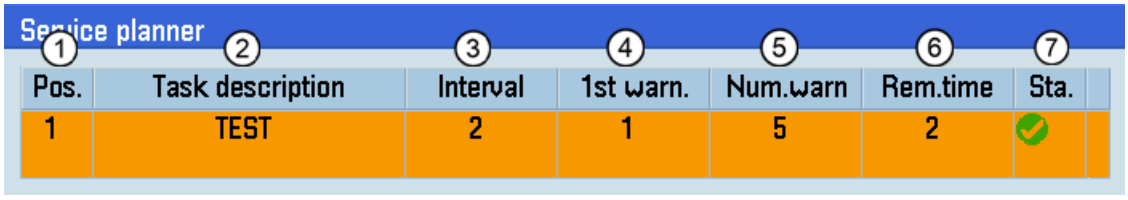
NOTICE
Shorter service life After NC restart, a future date set earlier than the lock date reduces the service life until the lock date. If a date equal to or later than the lock date is set, alarm 8064 is issued and the NC start disabled. Make sure you set the date correctly prior to NC restart.

3.12 Performing and monitoring maintenance tasks

You can open the service planner window through the following operations:



The service planner window displays a list of the maintenance tasks together with the position, task description, interval, first warning time, number of warnings, remaining time, and status.



① Position of the maintenance task in the PLC interface
② Name of the maintenance task
③ Maximum time until next servicing in hours
④ Time in hours at which an initial warning is displayed
⑤ Number of warnings that can be acknowledged by the operator before an alarm message is output for the last time

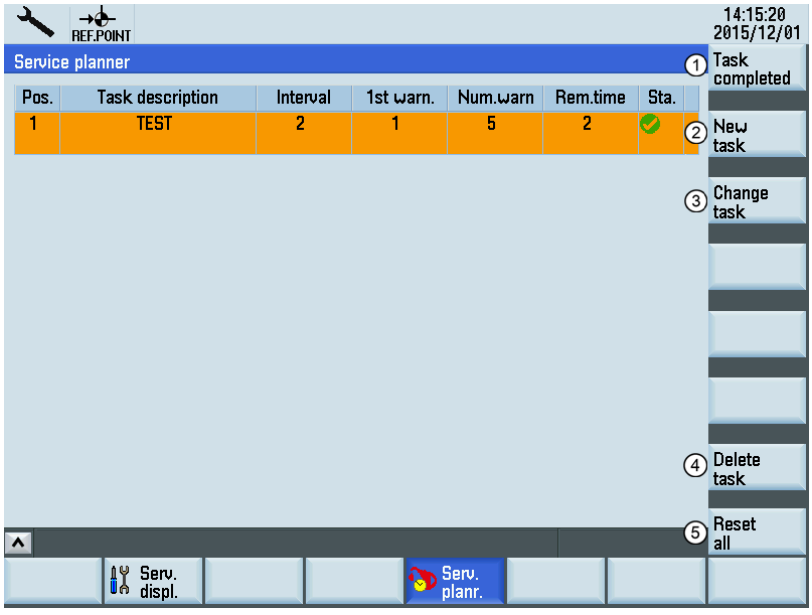
⑥ Time until the interval expires in hours. The remaining time cannot be edited.

⑦ Display of the current status of a maintenance task

: The maintenance task has been started.

: The maintenance task is completed.

In the service planner window, you can monitor the status of the maintenance tasks and acknowledge the tasks completed.



① Acknowledges the selected task which has been completed

② Creates a new task

③ Changes the selected task

④ Deletes the selected task

⑤ Resets the remaining time for all tasks

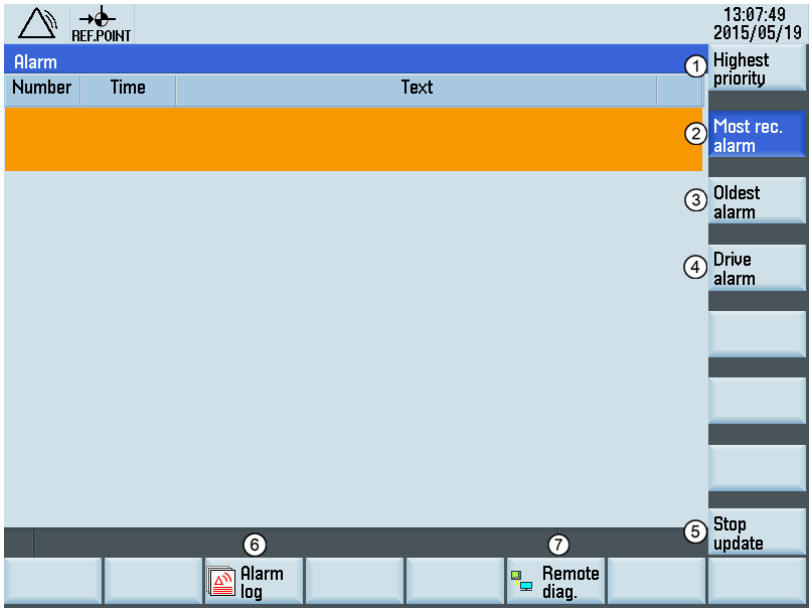
Note that softkeys ②, ③, ④, and ⑤ are accessible with the manufacturer or higher-level password.

Operating in the alarm operating area

4

Softkey functions

Pressing this key on the PPU allows you to enter the alarm operating area. You can check the NC and drive alarms using the softkeys. PLC alarms are not sorted.



① Displays all alarms sorted by their priorities. The highest priority alarm is at the beginning of the list.

② Displays the alarms sorted by the time of their occurrence. The most recent alarm stands at the beginning of the list.

③ Displays the alarms sorted by the time of their occurrence. The oldest alarm stands at the beginning of the list.

④ Displays the alarms on the drives. Note that this softkey is available on PPU16x.3 only.

⑤ Stops/starts updating of pending alarms

⑥ Views and manages the alarm log

⑦ Configures the access right for the remote control through the Ethernet connection. For more information about the softkey function, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

Viewing and managing alarm logs

1. Press this softkey to open the alarm log window.

2. Scroll up and down to view your desired alarm log. You can also sort the alarms by the time of their occurrence using the corresponding softkeys.

3. Press this softkey to save the log file to other storage media.

Pressing this softkey deletes all alarm logs.

SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms

5

5.1 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)

5.2 NCK alarms

To view detailed description about individual alarms directly on the PPU, proceed as follows:

1. Press this key on the PPU to enter the alarm operating area.

2. Select the desired alarm.

3. Press this key to open the online help for the selected alarm.

Note: You can further press this softkey in the current help screen to show a complete list of all SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms. In addition, you can also use the following softkey to search for a specific alarm by number in this list:

4. Pressing this softkey exits the help system.

2000 PLC sign-of-life monitoring

Explanation: 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: 100 ms).

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, please contact Technical Support.

www.siemens.com/sinumerik/help

Please supply the following information to ensure quick processing:

- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>

Programm continuation: Switch control OFF - ON.

2001 PLC has not started up

Explanation: The PLC must give at least 1 sign of life within a period of time defined in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT.

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.

If you experience such a system error, please contact Technical Support.

www.siemens.com/sinumerik/help

Please supply the following information to ensure quick processing:

- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>

Programm continuation: Switch control OFF - ON.

2110 NCK temperature alarm

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

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

Explanation: 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.

Programm 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)

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

2900 Reboot is delayed

Explanation: 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 canceled.

Remedy: See MD10088 $MN_REBOOT_DELAY_TIME and MD11410 $MN_SUPPRESS_ALARM_MASK

Programm continuation: Switch control OFF - ON.

3000 Emergency stop

Explanation: 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 canceled.

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 acknowledgment).

Programm continuation: Restart part program. 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

Explanation: 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.

Programm 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

Explanation: Only axes that have been activated in the channel by MD20070 $MC_AXCONF_MACHAX_USED [kx]=m may be declared as geometry axes, transformation axes or orientation axes in 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4006 The maximum number of activatable axes has been exceeded (limit %1)

Parameters: %1 = Number of axes

Explanation: 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).

Programm continuation: Switch control OFF - ON.

4009 Machine data %1%2 contains an illegal value.

Parameters: %1 = String: MD identifier
%2 = String: possibly further specification

Explanation: A value has been entered which lies above or below the value range or a limit value of 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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

Programm 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

Explanation: 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

Programm continuation: Switch control OFF - ON.

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

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

Explanation: The selected identifier is invalid. Valid identifiers are:
- AX1 - AXn: Machine axis identifiers
- N1AX1 - NnAXm: Link axis identifiers (NCU + machine axis), only for 'NCU-Link' expansion level!
- C1S1 - CnSm: Container axis identifiers (container + container location), only for 'Axis container' expansion level!

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

Remedy: Use the correct identifier.

Programm 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

Explanation: 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).

Programm continuation: Restart part program. 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

Explanation: 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).

Programm 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

Explanation: 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 turning machines), then channel axis 0 must be entered additionally in the channel-specific MD20050 $MC_AXCONF_GEOAX_ASSIGN_TAB.

Programm continuation: Switch control OFF - ON.

4032 [Channel %1:] Wrong identifier for facing axis in %2

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

Explanation: 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.

Programm 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.

Explanation: 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.
Only with active "OEM transformation" compile cycle: There are not enough channel axes entered in the MD displayed.

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.
Only with active "OEM transformation" compile cycle: In addition to the specified MD, check and correct MD24110 $MC_TRAFO_AXES_IN_1[n] of the activated OEM transformation according to the function description.

Programm 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

Explanation: 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 use of the specified machine data.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = String: Old identifier
%2 = String: New identifier

Explanation: 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.
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

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

4062 Backup data loaded

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

Reaction: Alarm display.

Remedy: Load specific machine data again.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Explanation: The user data of the NC and the retentive data of the PLC are stored in a buffered memory area (SRAM). A possible inconsistency was detected in this buffered memory during ramp-up of the control.

The buffered memory was initialized with the last backup copy. Changes that have been made since the last update of the backup copy in the buffered memory were thus lost.

A possible reason for a data inconsistency could be that the backup time was exceeded. Please note the required ON time of the control specified in your commissioning guide.

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

Remedy: Restart the control.

Programm continuation: Switch control OFF - ON.

4070 Normalizing machine data has been changed

Explanation: 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).

Programm continuation: Clear alarm with the Delete key or NC START.

4071 Check the position of the encoder

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4077 New value %1 of MD %2 not set. Requires %3 KB 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

Explanation: 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 required more user memory than was available.

The third parameter specifies the number of kilobytes by which the maximum user memory would be exceeded. Note: 1 KB = 1024 bytes. 1 MB = 1024 KB.

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 can be taken from MD18210 $MN_MM_USER_MEM_DYNAMIC, and is defined by OD19240 $ON_USER_MEM_DYNAMIC.

- "S" stands for static or buffered user memory (this is where part programs, offset data, R variables, tool data, etc. are typically stored). The capacity of this memory type can be taken from MD18210 $MN_MM_USER_MEM_BUFFERED, and is defined by OD19250 $ON_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 value of parameter 3.

- Is it possible to expand the memory? This option depends on the control model in use. (Never 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. An increase in buffered memory is subject to charge.

- Example: Parameter 4 is "D" - this means that $ON_USER_MEM_DYNAMIC should be increased. Parameter 3 is 400 kB - this means there is a lack of 400 / 1024 MiB = 1 MiB (rounded up). Increasing the value of $ON_USER_MEM_DYNAMIC by 1 MiB / 4 MiB = 1 (rounded up) makes the missing memory available.

Programm continuation: Clear alarm with the Delete key or NC START.

4080 Incorrect configuration of indexing axis in MD %1

Parameters: %1 = String: MD identifier

Explanation: 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.

Programm 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

Explanation: 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.
Programm continuation: Clear alarm with the RESET key. Restart part program

4090 Too many errors during power-up

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

4111 PLC cycle increased to %1 ms

Parameters: %1 = String (new PLC cycle time)
Explanation: The PLC cycle divisor was set to a value which was not an integral multiple of the IPO cycle divisor. The divisor (MD10074 $PLC_IPO_TIME_RATIO) has been increased.
For PROFIBUS/PROFINET: MD10074 $PLC_IPO_TIME_RATIO has been modified because of the modified DP cycle in the SDB (MD10050 $SYSCLOCK_CYCLE_TIME).
Reaction: Alarm display.
Remedy: MD10074 $MN_PLC_IPO_TIME_RATIO has been modified.
Programm continuation: Clear alarm with the RESET key. Restart part program

4115 Time ratio communication to Ipo changed to %1

Parameters: %1 = String (new ratio)
Explanation: 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.
Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4180 Invalid M function number assigned to enable ASUB

Explanation: An invalid M function number has been assigned for activation of ASUB. 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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number

Explanation: 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).

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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).

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number

Explanation: 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.

Programm 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

Explanation: 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 canceled.

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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = String: MD identifier

%2 = Index

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

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

Explanation: 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)

Programm continuation: Switch control OFF - ON.

4282 Hardware of external NCK outputs assigned repeatedly

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4340 [Channel %1:] Block %2 invalid transformation type in transformation no. %3

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

Explanation: An invalid, i.e. undefined number was entered in one of the machine data $MC_TRAFO_TYPE_.... This alarm also occurs if a certain type of transformation is only impossible on the type of control used (e.g. a 5-axis transformation).

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

Remedy: Enter a valid transformation type.

Programm continuation: Clear alarm with the RESET key. Restart part program

4341 [Channel %1:] Block %2 no data set available for transformation no. %3

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

Explanation: Only a limited number of machine data sets (usually 2) is available for each related group of transformations (e.g. orientation transformations, Transmit, Tracyl, etc.). This alarm is output if an attempt is made to set more transformations from a group.
Example:
Two orientation transformations are allowed. The machine data contains e.g.:
TRAFO_TYPE_1 = 16 ; 1st orientation transformation
TRAFO_TYPE_2 = 33 ; 2nd orientation transformation
TRAFO_TYPE_3 = 256 ; 1st transmit transformation
TRAFO_TYPE_4 = 20 ; 3rd orientation transformation ==> This entry triggers alarm

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

Remedy: Enter valid machine data.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = Channel number

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

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

Remedy: Set valid machine data.

Programm continuation: Clear alarm with the RESET key. Restart part program

4344 [Channel %1:] Block %2 Axis %3 defined in $NK_NAME[%4] is not available in the current channel

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name
%4 = Index of the chain element

Explanation: In the specified chain element a machine axis was indicated that is not available in the current channel on transformation selection. This means that this axis is currently assigned to another channel.

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

Remedy: Specify the axis available in the channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

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

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

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

4347 [Channel %1:] Invalid channel axis assignment in machine data %2[%3]

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

Explanation: MD2.... $MC_TRAFO_AXIS_IN_... contains an invalid entry. The following causes for the error are possible:
- The entry refers to a channel axis which does not exist.
- The entry is zero (no axis) but the transformation needs the relevant axis as a channel axis.
- More than one external axis has been entered in MD2.... $MC_TRAFO_AXIS_IN_... for the 7-axis transformation.

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

4348 [Channel %1:] Block %2 Configuration error %6 in transformation $NT_NAME[%5] = '%3'.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Name of the transformation data record
%4 = Index of the transformation data record | error number

Explanation: The transformation data set is incorrect. The cause of the error is given by the following error number:

- 1. The transformation type is not known, i.e. $NT_TRAFO_TYPE[n] contains an invalid name.
- 2. The machine kinematics have not been defined, i.e. neither in $NT_T_CHAIN_LAST_ELEM[n] nor in $NT_P_CHAIN_LAST_ELEM[n] is there a reference to a kinematic chain element in the kinematic description of the machine.
- 3. The kinematic chain element with the name contained in $NT_T_CHAIN_LAST_ELEM[n] has no connection to the root element.
- 4. The kinematic chain element with the name contained in $NT_P_CHAIN_LAST_ELEM[n] has no connection to the root element.
- 5. The kinematic chain element with the name contained in $NT_T_CHAIN_LAST_ELEM[n] was not found.
- 6. There is no kinematic chain element (MD18880 $MN_MM_MAXNUM_KIN_CHAIN_ELEM is zero).
- 8. The kinematic chain element with the name contained in $NT_P_CHAIN_LAST_ELEM[n] was not found.
- 9. The kinematic chain element with the name contained in $NT_T_REF_ELEM[n] was not found.
- 10. The rotary axis defined in $NT_ROT_AX_NAME[n,0] was not found in the kinematic chain.
- 11. The rotary axis defined in $NT_ROT_AX_NAME[n,1] was not found in the kinematic chain.
- 12. The rotary axis defined in $NT_ROT_AX_NAME[n,2] was not found in the kinematic chain.
- 16. The rotary axis defined in $NT_ROT_AX_NAME[n,1] has been defined more than once.
- 17. The rotary axis defined in $NT_ROT_AX_NAME[n,2] has been defined more than once.
- 20. The linear axis defined in $NT_GEO_AX_NAME[n,0] was not found in the kinematic chain.
- 21. The linear axis defined in $NT_GEO_AX_NAME[n,1] was not found in the kinematic chain.
- 22. The linear axis defined in $NT_GEO_AX_NAME[n,2] was not found in the kinematic chain.
- 26. The linear axis defined in $NT_GEO_AX_NAME[n,1] has been defined more than once.
- 27. The linear axis defined in $NT_GEO_AX_NAME[n,2] has been defined more than once.
- 30. The base orientation has not been defined, i.e. all three components of $NT_BASE_ORIENT[n, 0..2] are zero.
- 31. The orientation normal vector has not been defined, i.e. all three components of $NT_BASE_ORIENT_NORMAL[n, 0..2] are zero.
- 32. The vectors for definition of the base orientation ($NT_BASE_ORIENT[n, 0..2]) and the base normal vector ($NT_BASE_ORIENT_NORMAL[n, 0..2]) are parallel.
- 36. The number of relevant rotary axes found in the part chain is not the same as the content of $NT_ROT_AX_CNT[n, 0].
- 37. The number of relevant rotary axes found in the tool chain is not the same as the content of $NT_ROT_AX_CNT[n, 1].
- 40. The first and the second orientation axes of an orientation transformation are parallel.
- 41. The second and the third orientation axes of an orientation transformation are parallel.
- 42. No orientation axis has been defined (an orientation transformation requires at least one orientation axis).
- 43. Invalid 3-axis orientation transformation: The orientation axis is not perpendicular to the plane created by the two geometry axes.
- 47. The system variable $NT_CLOSE_CHAIN_P is disabled in the current software version, and may only contain the zero string.
- 48. The chain element specified in $NT_CLOSE_CHAIN_T was not found.
- 50. Fewer than two geometry axes have been defined for an orientation transformation or an inclined axis transformation.
- 51. Not all three geometry axes have been defined for an orientation transformation with more than one orientation axis.
- 60. Geometry axes 1 and 2 are parallel.
- 61. Geometry axes 1 and 3 are parallel.
- 62. Geometry axes 2 and 3 are parallel.
- 65. The 3 geometry axes are on one plane.
- 70. No geometry axis has been defined.
- 71. The only geometry axis defined must be in $NT_GEO_AX_NAME[n, 0].
- 72. One axis name must be entered in $NT_GEO_AX_NAME[n, 1].
- 73. More than 2 geometry axes have been defined.
- 74. One axis name must be in $NT_ROT_AX_NAME[n, 2], and only there.
- 75. The first geometry axis is not perpendicular to the rotary axis.
- 76. The second geometry axis is not parallel with the rotary axis.

- 77. The third linear axis is not perpendicular to the rotary axis and the first geometry axis.
- 78. More than one additional linear axis was defined.
- 79. Invalid kinematic chain element type (e.g. manual rotary axis).
- 80. Invalid definition sequence of orientation axes. The orientation axes in $NT_ROT_AX_NAME[n, 0..2] must be defined without gaps starting at index 0. The axis sequence in $NT_ROT_AX_NAME[n, 0..2] must be equal to the axis sequence in the kinematic chains (when running through the chains from the end of the part chain to the end of the tool chain).
- 81. An orientation axis has been programmed more than once.
- 82. Parameterization of the first orientation axis as a spindle is not permissible.
- 83. Parameterization of the second orientation axis as a spindle is not permissible.
- 84. Parameterization of the third orientation axis as a spindle is not permissible.
- 87. Parameterization of the first orientation axis as a Hirth axis is incorrect, i.e. at least one of the machine data MD30502 $MA_INDEX_AX_DENOMINATOR, MD30501 $MA_INDEX_AX_NUMERATOR, or MD30330 $MA_MODULO_RANGE (for modulo axes) is zero.
- 88. Parameterization of the second orientation axis as a Hirth axis is incorrect. The error conditions are the same as for error number 87.
- 89. Parameterization of the third orientation axis as a Hirth axis is incorrect. The error conditions are the same as for error number 87.
- 100. The maximum number of kinematic elements (sum of linear axes, rotary axes and constant elements) has been exceeded. In this case, a sequence of constant elements in a chain, which is not interrupted by an axis, is only considered as one element.

  A maximum of 15 kinematic elements is currently permissible for orientation transformations.
- 101. The maximum number of rotary axes for the kinematic chains for the definition of a transformation has been exceeded.
- 103. The maximum number of elements in the definition of the kinematic chain for the tool has been exceeded.
- 104. The maximum number of elements in the definition of the kinematic chain for the workpiece has been exceeded.
- 105. The maximum permissible number of constant transformation axes (redundant (rotary) axes), which with an active transformation must not move, has been exceeded.

  A maximum of 6 rotary axes is currently permissible for orientation transformations.
- 106. The maximum permissible number of chain elements for internal representation of the machine kinematics has been exceeded.
- 200. At the end of the part chain, a correction element is defined although bit 7 is set in $NT_CNTRL[n] (close part chain).
- 201. At the start of the tool chain, a correction element is defined although bit 8 is set in $NT_CNTRL[n] (close tool chain)
- 300. The chain element to which $NT_CORR_ELEM_P[n, 0] refers was not found.
- 301. The chain element to which $NT_CORR_ELEM_P[n, 1] refers was not found.
- 302. The chain element to which $NT_CORR_ELEM_P[n, 2] refers was not found.
- 303. The chain element to which $NT_CORR_ELEM_P[n, 3] refers was not found.
- 310. The chain element to which $NT_CORR_ELEM_T[n, 0] refers was not found.
- 311. The chain element to which $NT_CORR_ELEM_T[n, 1] refers was not found.
- 312. The chain element to which $NT_CORR_ELEM_T[n, 2] refers was not found.
- 313. The chain element to which $NT_CORR_ELEM_T[n, 3] refers was not found.
- 320. The chain element to which $NT_CORR_ELEM_P[n, 0] refers is not in the associated section.
- 321. The chain element to which $NT_CORR_ELEM_P[n, 1] refers is not in the associated section.
- 322. The chain element to which $NT_CORR_ELEM_P[n, 2] refers is not in the associated section.
- 323. The chain element to which $NT_CORR_ELEM_P[n, 3] refers is not in the associated section.
- 330. The chain element to which $NT_CORR_ELEM_T[n, 0] refers is not in the associated section.
- 331. The chain element to which $NT_CORR_ELEM_T[n, 1] refers is not in the associated section.
- 332. The chain element to which $NT_CORR_ELEM_T[n, 2] refers is not in the associated section.
- 333. The chain element to which $NT_CORR_ELEM_T[n, 3] refers is not in the associated section.
- 500. No transformation chain is defined for a concatenated transformation, at least one transformation must be defined with $NT_TRACON_CHAIN[].
- 501. The transformation stated in $NT_TRACON_CHAIN[n, 0] is not defined.
- 502. The transformation stated in $NT_TRACON_CHAIN[n, 1] is not defined.
- 503. The transformation stated in $NT_TRACON_CHAIN[n, 2] is not defined.
- 504. The transformation stated in $NT_TRACON_CHAIN[n, 3] is not defined.

- 520. The stated transformation data block is defective on activation of a concatenated transformation.
- 521. The definition of the transformation chain leads to an endless recursion.
- 522. Too many concatenated transformations are defined for the definition of a transformation chain (8 is the maximum possible here).
- 523. An impermissible chain of partial transformations has been configured in a concatenated transformation.
- 1000. The OEM transformation cannot be activated because it was not logged on by the CC.
- 1001. The OEM transformation has been logged on as an orientation transformation, but it is not configured as an orientation transformation.
- 1002. The OEM transformation is configured as an orientation transformation, but it was not logged on as an orientation transformation.
- 1003. The activated OEM transformation does not behave like an orientation transformation, although the transformation was not configured or logged on for orientations.
- 1004. No geometry axes are defined for the activated OEM transformation (neither by the CC nor in the system variable $NT_GEO_AX_NAME[n, i]).
- 1005. No orientation axes are defined for the activated OEM transformation, although this was configured as an orientation transformation (neither by the CC nor in the system variable $NT_ROT_AX_NAME[n, i]).
- 1006. The OEM transformation does not support any rotation of the orientation, although the transformation was configured with 3 orientation axes.
- 1007. The OEM transformation was created for rotations of the orientation, but the transformation was not configured in this way (fewer than 3 orientation axes).
- 1050. The OEM transformation reports a general kinematic analysis error. (This occurs, for example, if the analyseChain() method has not been implemented in the CC.)
- 1051. An unknown element was discovered during the analysis of the kinematic chain by the OEM transformation.
- 1052. An offset element is incorrectly arranged in the kinematic chain for the OEM transformation.
- 1053. The stated arrangement of a linear axis is not allowed in the kinematic chain.
- 1054. The stated arrangement of a rotary axis is not allowed in the kinematic chain.
- 1055. The stated direction of a linear axis is not allowed in the kinematic chain.
- 1056. The stated direction of rotation of a rotary axis is not allowed in the kinematic chain.
- 1057. Reserved for error messages of the kinematic analysis of compile cycles.
- 1058. Reserved for error messages of the kinematic analysis of compile cycles.
- 1059. Reserved for error messages of the kinematic analysis of compile cycles.
- 1060. Reserved for error messages of the kinematic analysis of compile cycles.
- 1061. Reserved for error messages of the kinematic analysis of compile cycles.
- 1062. Reserved for error messages of the kinematic analysis of compile cycles.
- 1063. Reserved for error messages of the kinematic analysis of compile cycles.
- 1064. Reserved for error messages of the kinematic analysis of compile cycles.
- 1065. Reserved for error messages of the kinematic analysis of compile cycles.
- 1066. Reserved for error messages of the kinematic analysis of compile cycles.
- 1067. Reserved for error messages of the kinematic analysis of compile cycles.
- 1068. Reserved for error messages of the kinematic analysis of compile cycles.
- 1069. Reserved for error messages of the kinematic analysis of compile cycles.
- 1070. The orientation of the tool in the basic position must be parallel to one of the coordinate axes X, Y or Z (value of the variable $NT_BASE_ORIENT[i]).
- 1071. The orientation of the tool in the basic position must not be parallel to the X direction (value of the variable $NT_BASE_ORIENT[0]).
- 1072. The orientation of the tool in the basic position must not be parallel to the Y direction (value of the variable $NT_BASE_ORIENT[1]).
- 1073. The orientation of the tool in the basic position must not be parallel to the Z direction (value of the variable $NT_BASE_ORIENT[2]).
- 1074. The normal vector of the tool in the basic position must be parallel to one of the coordinate axes X, Y or Z (value of the variable $NT_BASE_ORIENT_NORMAL[i]).
- 1075. The normal vector of the tool in the basic position must not be parallel to the X direction (value of the variable $NT_BASE_ORIENT_NORMAL[0]).
- 1076. The normal vector of the tool in the basic position must not be parallel to the Y direction (value of the variable $NT_BASE_ORIENT_NORMAL[1]).

- 1077. The normal vector of the tool in the basic position must not be parallel to the Z direction (value of the variable $NT_BASE_ORIENT_NORMAL[2]).
- 1078. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1079. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1080. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1081. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1082. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1083. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1084. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1085. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1086. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1087. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1088. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1089. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 1090. Reserved for error messages of the analysis of the transformation parameters of compile cycles.
- 10000. Invalid redundant rotary axis. Only one redundant rotary axis is permissible for orientation transformation (for the time being). This rotary axis must be the first axis in the kinematic chain.

Note:

Parameter 4 contains the descriptions of parameters 5 and 6 separated by the character "|".

- 5 = Index of the transformation data block
- 6 = Error code

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

Remedy: Define valid transformation data record.

Programm continuation: Clear alarm with the RESET key. Restart part program

4349 [Channel %1:] No free memory space available for transformations.

Parameters: %1 = Channel number
%2 = Number of the transformations already active

Explanation: Any kinematic transformation in the NCK requires a defined memory space. If MD18866 $MN_MM_NUM_KIN_TRAFOS does not equal zero, it indicates how many kinematic transformations are allowed to be active in the NCK at the same time.

If MD18866 $MN_MM_NUM_KIN_TRAFOS equals zero, the maximum number of kinematic transformations that are active at the same time, is determined automatically (currently twenty times the number of the existing channels).

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

Remedy: Increase the value of MD18866 $MN_MM_NUM_KIN_TRAFOS.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

4402 %1 causes a machine data reset

Parameters: %1 = Machine data

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

4500 Error %2 occurred (data loss!) while writing AMR backup file %1.

Parameters: %1 = File name

%2 = Error code

Explanation: The backup file for the automatic memory configuration was not able to be written to

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Switch control OFF - ON.

4501 While reading the AMR backup file %1, error %2 occurred (data loss!)

Parameters: %1 = File name

%2 = Error code

Explanation: The backup file for automatic memory reconfiguration could not be read

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number

%2 = String: MD identifier

%3 = String: MD identifier

%4 = String: MD identifier

Explanation: 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: --

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

4600 Invalid handwheel type for handwheel %1

Parameters: %1 = Handwheel number

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4610 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4611 Invalid handwheel input for handwheel %1

Parameters: %1 = Handwheel input

Explanation: 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 8xxD systems.

Reaction: Interface signals are set.
Alarm display.

Remedy: Configure MD11352 $MN_HANDWHEEL_INPUT for a valid input for the corresponding handwheel

Programm continuation: Switch control OFF - ON.

4620 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4621 Invalid handwheel input for handwheel %1

Parameters: %1 = Handwheel input

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4630 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Explanation: For PROFIBUS/PROFINET only:
The reference in $MN_HANDWHEEL_MODULE to a corresponding entry in machine data array $MN_HANDWHEEL_LOGIC_ADDRESS[] which is required for configuring PROFIBUS handwheels is not available.

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[].

Programm continuation: Switch control OFF - ON.

4631 Invalid handwheel slot for handwheel %1

Parameters: %1 = Handwheel slot

Explanation: For PROFIBUS/PROFINET only:
The handwheel slot for handwheel %1 requested through machine data $MN_HANDWHEEL_INPUT is not available for PROFIBUS handwheels.

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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Handwheel number

Explanation: For PROFIBUS/PROFINET only:
The logical basic address of the PROFIBUS handwheel slot in machine data array $MN_HANDWHEEL_LOGIC_ADDRESS[] indexed in machine data $MN_HANDWHEEL_MODULE was not found in the current STEP 7 hardware configuration.

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.

Programm continuation: Switch control OFF - ON.

4640 Invalid handwheel module for handwheel %1

Parameters: %1 = Handwheel module

Explanation: 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.

Programm continuation: Switch control OFF - ON.

4641 Invalid handwheel input for handwheel %1

Parameters: %1 = Handwheel input

Explanation: 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

Programm continuation: Switch control OFF - ON.

5000 Communication job not executable %1

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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6000 Memory reorganized using standard machine data

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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 areas, 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 areas, channel-specific
- _N_CHc_UFR - System internal: frames, channel-specific
- _N_CHc_RPA - System internal: R-variables, 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 areas, 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.

Programm continuation: Switch control OFF - ON.

6020 Machine data have been changed - now memory %1 is reorganized

Parameters: %1 = Detailed information

Explanation: 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.

Meaning of the parameter value:

- AFS = The active file system was reconfigured. The files of the passive file system were retained.
- PFS/AFS = The passive and active file systems were reconfigured.

Via the machine data $MN_IS_AUTOMATIC_MEM_RECONFIG it is configured whether NCK configures the AFS automatically (value=TRUE) or not (value=FALSE).

Reaction: Alarm display.

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

Programm continuation: Clear alarm with the RESET key. Restart part program

6030 Limit of user memory has been adapted

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = TO unit

%2 = Tool identifier (name)

%3 = -Not used-

%4 = D number

Explanation: Tool monitoring: This message informs that the specified D offset has reached its prewarning limit for a time-, quantity- or wear-monitored tool. If possible, the D number is displayed; if not, value 0 is assigned to the 4th parameter.

If the function additive offset is being used, additive offset monitoring may be active instead of tool wear monitoring. The actual type of tool monitoring is a tool property (see $TC_TP9). If replacement tools are not being used, the duplo number specified has no meaning. The alarm is triggered through the HMI or PLC (=OPI interface). The channel context is not defined. The TO unit was specified for this reason (see MD28085 $MC_MM_LINK_TOA_UNIT).

Reaction: Interface signals are set.

Alarm display.

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

Programm continuation: Clear alarm with the Delete key or NC START.

6411 [Channel %1:] Tool %2 has reached its prewarning limit with D = %4

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

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

Reaction: Interface signals are set.
Alarm display.

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

Programm continuation: Clear alarm with the Delete key or NC START.

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

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

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

Reaction: Interface signals are set.
Alarm display.

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

Programm continuation: Clear alarm with the Delete key or NC START.

6413 [Channel %1:] Tool %2 has reached its monitoring limit with D = %4

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

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

Reaction: Interface signals are set.
Alarm display.

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

Programm continuation: Clear alarm with the Delete key or NC START.

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

Explanation: No more cutting edges can be entered in the piece counter table.

As many cutting edges can be noted for the workpiece counter as are possible in total in the NCK.

This means that if for each tool each cutting edge in each TO unit is used precisely once for a workpiece then the limit is reached.

If several workpieces are made on several toolholders/spindles simultaneously, it is possible to note

MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA cutting edges for the workpiece counter for all of the workpieces.

If this alarm occurs, it means that cutting edges used subsequently are no longer quantity monitored until the table has been emptied again, e.g. by means of the NC language command SETPIECE or by the relevant job from HMI, PLC (PI service).

Reaction: NC Start disable in this channel.

Interface signals are set.

Alarm display.

Remedy: - Was decrementing of the piece counter forgotten? Then program SETPIECE in the part program, or add the correct command in the PLC program.

- If the part program/PLC program is correct, then more memory should be set for tool cutting edges via the MD18100 $MN_MM_NUM_CUTTING_EDGES_IN_TOA (can only be performed with the necessary access rights!).

Programm continuation: Clear alarm with the Delete key or NC START.

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

Parameters: %1 = Channel ID

%2 = Block number, label

Explanation: 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.

Programm 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

Explanation: The command cannot be programmed due to the lack of a function enable or activation.

Function code (4th parameter):

1 = Flat D numbers

2 = Tool monitoring

3 = Magazine management

4 = Multitools

5 = T=Magazine location number

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

Remedy: Correct the NC program

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

6437 [Channel %1:] Block %2 Command '%3' cannot be programmed. Function '%4' is activated.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Programmed command
%4 = Function identifier

Explanation: The command cannot be programmed as the specified function is active.
Function code (4th parameter):
1 = Flat D numbers
2 = Tool monitoring
3 = Magazine management
4 = Multitools
5 = T=Magazine location number

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

Remedy: Correct the NC program

Programm 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

Explanation: 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).

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

6500 NC memory full

Explanation: 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, HMI files, FIFO files, NC programs...

Reaction: Alarm display.

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

Programm continuation: Clear alarm with the Delete key or NC START.

6510 Too many part programs in the NC memory

Explanation: 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, HMI files, FIFO files, NC programs, cycle 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 or MD18321 MD_MAXNUM_SYSTEM_FILES_IN_FILESYSTEM.
- With the Siemens cycle storage, increase the number of files in the ressource file.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6530 Too many files in directory

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6540 Too many directories in the NC memory

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6550 Too many subdirectories

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6560 Data format not allowed

Explanation: 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).

Programm continuation: Clear alarm with the Delete key or NC START.

6568 Limit of the extended CNC memory reached

Explanation: The memory space assigned to the function 'Extended CNC user memory' is exhausted.

Reaction: NC Start disable in this channel.

Alarm display.

Remedy: Delete files in the extended CNC memory

Programm continuation: Switch control OFF - ON.

6569 HMI user memory limit reached on the CF card.

Explanation: The memory space in the user area assured on the CF card is exhausted.

Reaction: NC Start disable in this channel.

Alarm display.

Remedy: Delete files on the CF card

Programm continuation: Switch control OFF - ON.

6570 NC memory full

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

6580 NC memory full

Explanation: 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. part programs)

Programm continuation: Clear alarm with the Delete key or NC START.

6581 NC user memory full

Explanation: 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. part programs)

Programm continuation: Clear alarm with the Delete key or NC START.

6582 NC machine OEM memory full

Explanation: 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. part programs)
Programm continuation: Clear alarm with the Delete key or NC START.

6583 NC system memory full

Explanation: The DRAM file system of the system area (Siemens) is full. The order cannot be executed.
Reaction: Alarm display.
Remedy: Increase MD18354 $MN_MM_S_FILE_MEM_SIZE or, with Siemens cycles, increase the partition size in the cycle ressource file.
Programm continuation: Clear alarm with the Delete key or NC START.

6584 NC memory limit TMP reached

Explanation: 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.
Programm continuation: Clear alarm with the Delete key or NC START.

6585 NC external memory limit reached

Explanation: 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.
Programm continuation: Clear alarm with the Delete key or NC START.

6693 File %1 lost

Parameters: %1 = File name
Explanation: 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.
Programm continuation: Switch control OFF - ON.

6694 Drives could not be mounted

Explanation: The external drives could not be mounted within the configured time (see MD10128 $MN_EES_MAX_MOUNT_TIME).
Reaction: NC not ready.
Interface signals are set.
Alarm display.
Remedy: Check logdrives.ini and increase MD10128 $MN_EES_MAX_MOUNT_TIME if necessary.
Programm continuation: Clear alarm with the RESET key. Restart part program

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

Explanation: 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.

Programm 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

Explanation: On assigning a protection level for a part program command via REDEF command
- an impermissible part 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 part 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = Channel number

Explanation: Fast NC 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!

Programm 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

Explanation: 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 authorised 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.
Reduce the number of active axes or enable additional axes via option.
Please also observe the information relating to the definition of 'auxiliary axes/spindles', 'simulation axes', and 'virtual axis for master value coupling with curve table'.

Programm continuation: Switch control OFF - ON.

8012 The 'Activation of more than %1 SI axes' option is not set

Parameters: %1 = Number of licensed axes
%2 = Number of axes with active dbSi

Explanation: The drive-based safety functions were activated for more axes than permitted 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 notify the authorized personnel / service department. Reduce the number of axes activated with drive-based safety functions or release additional axes in the option date.

Programm continuation: Switch control OFF - ON.

8022 Option 'activation of more than %1KB SRAM' not set

Parameters: %1 = Memory size

Explanation: 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

Programm continuation: Switch control OFF - ON.

8025 [Channel %1:] Option '%2<OPTNX>' not set

Parameters: %1 = Channel number
%2 = Brief description of option

Explanation: 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)

Programm continuation: Switch control OFF - ON.

8027 '%1<OPTNX>' option not set

Parameters: %1 = Brief description of option

Explanation: The option for the 'Grinding Advanced' functionality is not set.

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

Remedy: Please contact the authorized personnel/Service.
- Buy option
- Deactivate the 'Grinding Advanced' functionality
- 'Grinding Advanced' functional scope:
  Cylinder error compensation

Programm 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

Explanation: The number of axes programmed in the interpolating group exceeds the permissible number of interpolating axes.

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

Remedy: In the part program, program as many axes as are permitted by the configuration of the controller or enable additional axes via option.

Programm 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

Explanation: 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

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = String: MD identifier

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

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

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

Explanation: The number of axes specified in the associated option data has been exceeded.
More axes have been selected in the corresponding machine data than are permitted by the function associated with the option.
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: Please inform the authorized personnel/service department.
- Increase option data
- Reduce number of axes

Programm continuation: Switch control OFF - ON.

8042 Option '%3<OPTNX>' is not set

Parameters: %1 = Brief description of option

Explanation: A non-released option was used
The stated option or an equivalent one is required for execution of the action.

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

Remedy: Terminate action, upgrade option.
Please compare the available option data and/or (if available) the license image of your controller

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

8045 'Activation drive integrated ESR' option not set

Explanation: 'Extended stop and retract (ESR)' was activated in the drive, although the corresponding option was not 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: - Purchase option
- Deactivate 'Extended stop and retract (ESR)' function in the drive.

Programm continuation: Switch control OFF - ON.

8052 Drive: License %1<DRVLICPANX> with the value %2 is invalid

Parameters: %1 = Drive, license ID/MLFB
%2 = Drive, license value

Explanation: The drive option has not been enabled, or the number of enabled drive options was exceeded

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

Remedy: Please notify the authorized personnel/service.
- Reset drive option
- Purchase option

Programm continuation: Switch control OFF - ON.

8053 Drive: License %1 is unknown

Parameters: %1 = Drive, option ID

Explanation: Drive option is unknown

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

Remedy: Please notify the authorized personnel/service.
- Reset drive option

Programm continuation: Switch control OFF - ON.

8060 CNC lock function: The request cannot be processed: cause %1

Parameters: %1 = Cause

Explanation: The start conditions of the CNC lock function are not met.
The following cause was detected:
1 - The clock is not set.
2 - The option bit was deactivated.
3 - The control type is not supported.
4 - The PLC project is not supported.
5 - No license is available.
6 - MD17300 $MN_CNC_LOCK_WARNING_TIME is zero.
7 - The previous order is still being processed.
11 - The LockSet file was not found.
12 - The LockSet file cannot be read.
13 - The version of the LockSet file is not supported.
14 - The number of the CF card is unknown.
15 - The lock date is invalid.
16 - The hardware serial number is unknown.
31 - The current mode is not permitted.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Switch control OFF - ON.

8061 CNC lock function cannot be updated : cause %1 .

Parameters: %1 = Cause

Explanation: The desired update of the active CNC lock function is rejected.
The following cause was detected:
1 - The clock is not set.
2 - The option bit had been deactivated.
4 - The PLC project is not supported.
5 - No license is available.
6 - MD17300 $MN_CNC_LOCK_WARNING_TIME is zero.
7 - The previous order is still being processed.
11 - The LockSet file was not found.
12 - The LockSet file cannot be read.
13 - The version of the LockSet file is not supported.
14 - The number of the CF card is unknown.
15 - The lock date is invalid.
16 - The hardware serial number is unknown.
21 - The OEM key is incorrect.
22 - The PLC project was replaced.
31 - The current mode is not permitted.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Switch control OFF - ON.

8062 CNC lock function: The execution of the function was interrupted: cause %1_

Parameters: %1 = Cause

Explanation: The execution of the already active CNC lock function has been modified.
The following cause was detected:
1 - The CF card was exchanged.
2 - The hardware was exchanged.
3 - The PLC project was changed.
4 - The PLC reports an incorrect OEM key.
5 - The data of the function was subject to unauthorized change.
6 - The PLC project is not supported.
The NC start is locked.

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

Remedy: Please restore the former status of the control.
If this cannot be done, please inform the authorised personnel/service department, and reactivate the CNC lock function.

Programm continuation: Switch control OFF - ON.

8063 CNC lock function: The lock date will be reached in less than %1 days!

Parameters: %1 = Number of production days remaining

Explanation: A reminder is issued for the lock date in accordance with the agreements for operating this machine configuration.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Clear alarm with the Delete key or NC START.

8064 CNC lock function: The lock date has been reached, an NC start is not possible!

Explanation: The lock date has been reached in accordance with the agreements for operating this machine configuration.
NC start is locked.

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

Remedy: Please inform the authorized personnel/service department.

Programm continuation: Switch control OFF - ON.

8065 CNC lock function: Please set the correct date/time!

Explanation: While the CNC lock function is active, incorrect settings for date/time have been found.
NC start is locked.
The time of day must be corrected without fail before the control is switched off. Otherwise there could be a permanent lock caused by alarm 8064 "CNC lock function: The lock date has been reached, and NC start is not possible!".

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

Remedy: Please set the correct date/time.

Programm continuation: Switch control OFF - ON.

8066 CNC lock function: Changing the date has reduced the length of the remaining runtime.

Explanation: Activating the CNC lock function has brought forward the date of the control. This has reduced the length of time before the lock date is reached.

Reaction: Alarm display.
Remedy: The date of the control can be reset before switching off/on.
Programm continuation: Clear alarm with the Delete key or NC START.

8067 CNC lock function: A system alarm has occurred: cause %1

Parameters: %1 = Cause
Explanation: The CNC lock function has found problems:
1 - Internal interface 1
X - Internal interface X
Reaction: NC not ready.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: If you experience such a system error, please contact Technical Support.
www.siemens.com/sinumerik/help
Please supply the following information to ensure quick processing:
- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>
Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Number of non-licensed options
Explanation: 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/license and enter it in the operating area "Setup", function (HSK) "Licenses".
Programm 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
Explanation: 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/license and enter it in the operating area "Setup", function (HSK) "Licenses"..
Programm 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.

Explanation: 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).
Programm continuation: Clear alarm with the Delete key or NC START.

8083 Export-restricted system software without valid licensing.

Explanation: 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/license, and enter the license key under the "Start-up" operating area, function (HSK) "Licenses".
Note:
If this alarm occurs after the replacement of a defective CompactFlash card, please observe the following:
On account of export regulations applicable to the SINUMERIK 840D sl, only CompactFlash cards with the following MLBFs may be used to replace a defective CompactFlash card:
6FC5851-1XG##-#YA0
6FC5851-1XG##-#YA8
There are already license keys on these CompactFlash cards that enable export-restricted system software to be installed and operated.
Back up this license key before overwriting the CompactFlash card with a software image, and then restore the key.
If you forget to make the backup, you can restore the display of the original license key of your CompactFlash card via http://www.siemens.com/automation/license.

Programm continuation: Switch control OFF - ON.

8084 Period %1 of the test license active remaining time %2 h

Explanation: A test period has been started. During this period, one or several options can be set, which require a license key to prove that they have been purchased
A maximum of 6 test periods are possible, one test period can be a maximum of 150 h (up to 3000 h for larger control models) operating time of the control system

Reaction: Alarm display.

Remedy: Generate a new license key through the Internet at http://www.siemens.com/automation/license and enter in the operating area "Startup", function (HSK) "Licenses".
Enter a valid license key in the operating area "Startup", function (HSK) "Licenses".

Programm continuation: Clear alarm with the Delete key or NC START.

8085 %1. Test license period has expired

Explanation: A test period has expired.

Reaction: Alarm display.

Remedy: Generate a new license key via the Internet at http://www.siemens.com/automation/license and enter in the operating area "Startup", function (HSK) "Licenses".
Enter a valid license key in the operating area "Startup", function (HSK) "Licenses".
Activate an additional test period
Reset the options that require a license key

Programm continuation: Clear alarm with the Delete key or NC START.

8086 Test and demonstration machine

Explanation: This controller is operated with a test license for testing and demonstration purposes.
For licensing reasons, you may not use this machine in the productive process.

Reaction: Alarm display.

Remedy: Generate a new license key through the Internet at http://www.siemens.com/automation/license and enter in the operating area "Startup", function (HSK) "Licenses".
Enter a valid license key in the operating area "Startup", function (HSK) "Licenses".
Programm continuation: Clear alarm with the Delete key or NC START.

8089 The function 'Select tool offsets' is not possible

Explanation: The current options do not permit a tool offset.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Disable tools offsets or
reset option packages that include the tool offset as a reduced function.
Programm continuation: Clear alarm with the RESET key. Restart part program

8090 CNC lock function: No license available

Explanation: The CNC lock function is not licensed.
A test license is not acceptable.
Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: Generate a new license key on the internet at http://www.siemens.com/automation/license and enter it in the operating area "Setup", function (HSK) "Licenses"..
Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: - 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.
Programm 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

Explanation: - 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

9000 Handwheel %1 failed

Parameters: %1 = Handwheel number

Explanation: For PROFIBUS/PROFINET only:
PROFIBUS handwheel has failed

Reaction: Interface signals are set.
Alarm display.

Remedy: Restore connection to PROFIBUS handwheel

Programm 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

Explanation: 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).

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10204 [Channel %1:] User action not possible without reference point (internal action=%2<ALNX>)

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

Explanation: A specific user action is to be executed, which leads to a (possibly other) internal action, for which as least one axis that needs to be referenced has not reached its reference point.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please reference the axis that needs to be referenced.

If the user action as an internal action leads to an NC start, whether or not the axis has to be referenced before NC start can be decided via the channel-specific MD20700 $MC_REFP_NC_START_LOCK (NC start without reference point). If the user action as internal actions leads to a user ASUB start, whether or not the axis has to be referenced before ASUB start can be decided via the channel-specific MD20115 $MC_IGNORE_REFP_LOCK_ASUP (ASUB start without reference point). If the user action as internal actions leads to a Prog Event start, whether or not the axis has to be referenced before Prog Event start can be decided via the channel-specific MD20105 $MC_PROG_EVENT_IGN_REFP_LOCK (Prog Event start without reference point).

Channel or axis-specific referencing can be triggered.

Please inform the authorized personnel/service department. Whether or not the axis has to be referenced before NC start can be determined via the channel-specific MD20700: $MC_REFP_NC_START_LOCK (NC start without reference point). Channel or axis-specific referencing can be triggered.

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).

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = Channel number

Explanation: 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.

Programm 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

Explanation: 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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10225 [Channel %1:] command denied

Parameters: %1 = Channel number

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

Reaction: Alarm display.

Remedy: Press RESET.

Programm continuation: Clear alarm with the Delete key or NC START.

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

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

Parameters: %1 = Channel number

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

Reaction: Alarm display.

Remedy: This message is purely informational.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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 canceled.

Programm 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.

Explanation: 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).

Programm 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 -)

Explanation: 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.

Programm 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.

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

10632 [Channel %1:] Block %2 axis %3 reaches the coordinate system-specific working area limit %4

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

Explanation: The specified axis violates the coordinate system-specific working area limitation. This is not detected until the main run, either because the minimum axis values could not be determined before the transformation or because there is an overlaid movement.

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

10633 [Channel %1:] Axis %2 is at coordinate system-specific working area limit %3%4

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

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

Reaction: Alarm display.

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

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

10634 [Channel %1:] Axis %2, tool radius compensation is inactive for type %3 working area limitation, reason: The tool is not oriented parallel to the axis.

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = 0: BCS, 1: WCS / SZS

Explanation: The tool radius compensation of the working area limitation of the stated axis is not taken into account.
Reason: The tool is not oriented parallel to the axis (e.g. because toolcarrier or transformation is active).
The alarm is reported in JOG mode.

Reaction: Alarm display.

Remedy: The tool radius compensation for working area limitations in JOG mode can only be taken into account if the tool is parallel to the axis.
Active transformation and toolcarrier must be switched off for this function.

Programm continuation: Clear alarm with the Delete key or NC START.

10635 [Channel %1:] Axis %2, tool radius compensation is inactive for type %3 working area limitation, reason: no milling or drilling tool.

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = 0: BCS, 1: WCS / SZS

Explanation: The tool radius compensation of the working area limitation of the stated axis is not taken into account.
Reason: The tool must be of type milling cutter or drill.
The alarm is reported in JOG mode.

Reaction: Alarm display.

Remedy: The tool radius compensation for working area limitations in JOG mode can only be taken into account for milling or drilling tools.

Programm continuation: Clear alarm with the Delete key or NC START.

10636 [Channel %1:] Axis %2, tool radius compensation is inactive for type %3 working area limitation, reason: Transformation is active.

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = 0: BCS, 1: WCS / SZS

Explanation: The tool radius compensation of the working area limitation of the stated axis is not taken into account.
Reason: A transformation is active.
The alarm is reported in JOG mode.

Reaction: Alarm display.

Remedy: The tool radius compensation for working area limitations in JOG mode cannot be taken into account if transfomation is active.

Programm continuation: Clear alarm with the Delete key or NC START.

10637 [Channel %1:] Axis %2, tool radius compensation is inactive for type %3 working area limitation, reason: Tool not active.

Parameters: %1 = Channel number
%2 = Axis, spindle
%3 = 0: BCS, 1: WCS / SZS

Explanation: The tool radius compensation of the working area limitation of the stated axis is not taken into account.
Reason: No tool is active.
The alarm is reported in JOG mode.

Reaction: Alarm display.

Remedy: The tool radius compensation for working area limitations in JOG mode cannot be taken into account wihout an active tool.

Programm continuation: Clear alarm with the Delete key or NC START.

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

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

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

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

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

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number
%2 = Reason

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

e.g. error code 1001 = no master axis declared, gantry unit 1.

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

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

Programm continuation: Switch control OFF - ON.

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

Parameters: %1 = Channel number
%2 = Axis

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

Reaction: Alarm display.

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

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters: %1 = Channel number
%2 = Axis

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

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

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

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

Explanation: The alarm message appears when the axes are ready for synchronization. The gantry grouping can now be synchronized. The actual value difference between the master and slave axes is greater than the gantry warning limit MD 37110 $MA_GANTRY_POS_TOL_WARNING. The synchronization must be started explicitly with the NC/PLC interface signal DB380x DBX5005.4 (Start gantry synchronization).

Reaction: Alarm display.

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

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

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

Explanation: The alarm may be suppressed with MD37150 $MA_GANTRY_FUNCTION_MASK Bit2 = 1.

Reaction: Alarm display.

Remedy: --

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters: %1 = Channel number
%2 = Axis

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

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

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

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

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

Parameters: %1 = Channel number
%2 = Axis

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

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

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

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

Explanation: Error ID and gantry unit
- 30XX => Gantry group cannot be closed, as not all gantry axes are in one channel.
- 40XX => Gantry group cannot be closed, as the gantry axes have different axis states (the axis is assigned to the PLC, for example).
- 50XX => Gantry group is to change channel due to a PLC request, not all gantry axes are known in the new channel.
- 60XX => Gantry group is to be transferred to the channel due to an NC program request, but the channel does not know all the gantry axes.
- 70XX => Gantry group cannot be closed, as movement is pending for at least one gantry axis.

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

10700 [Channel %1:] Block %2 NCK protection area %3 violated during automatic or MDI mode

Parameters: %1 = Channel number
%2 = Block number
%3 = Protection area number

Explanation: The workpiece-related NCK protection area has been violated. Note that another tool-related protection area 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.
The SHOWALARM and SETVDI reactions can be suppressed with MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit20.

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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10701 [Channel %1:] Block %2 channel-specific protection area %3 violated during automatic or MDI mode

Parameters: %1 = Channel number
%2 = Block number
%3 = Protection zone number

Explanation: The workpiece-related channel-specific protection area has been violated. Note that another tool-related protection area 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.
The SHOWALARM and SETVDI reactions can be suppressed with MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit20.

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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10702 [Channel %1:] NCK protection area %2 violated during manual mode

Parameters: %1 = Channel number
%2 = Protection zone number

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters: %1 = Channel number
%2 = Protection zone number

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

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

Explanation: 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 areas 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 areas. (The warning comes nevertheless) or exclude additional motions.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters: %1 = Channel number
%2 = Protection zone number
%3 = Axis name

Explanation: The workpiece-related NCK protection area has been reached with the specified axis. Note that another tool-related protection area is still active. The workpiece-related protection area 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. The protection area can be traversed after enable signal from PLC.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

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

Parameters: %1 = Channel number
%2 = Protection zone number
%3 = Axis name

Explanation: The workpiece-related channel-specific protection area has been reached with the specified axis. Note that another tool-related protection area is still active. The workpiece-related protection area 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.

Programm 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 -)

Explanation: 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.

Programm 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 -)

Explanation: 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.

Programm 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

Explanation: This alarm was triggered during the preparation of the displayed block.
Cause:
The programmed path violates the active software limit switch in the traversing direction for the displayed axis.
Note:
Parameter 4 contains the descriptions of parameters 5, 6 and 7 separated by the character "|".
- 5 = Block number, label
- 6 = Number + string (+/-)
- 7 = Distance-to-go

Alarm 10722 is displayed for MD11411 $MN_ENABLE_ALARM_MASK, bit11 == 1 instead of alarm 10720.
Alarm 10722 offers an expanded diagnostics option for the software limit switch violation.
Condition:
Alarm file ALUN* is present in the HMI.
See also:
Diagnostics Manual for alarm 10720

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

Remedy: Check and, if necessary, correct the positions programmed in the NC program.
Please inform the authorised personnel/service department.
Check the parameterisation of the software limit switch:
- MD36100 $MA_POS_LIMIT_MINUS
- MD36120 $MA_POS_LIMIT_MINUS2
- MD36110 $MA_POS_LIMIT_PLUS
- MD36130 $MA_POS_LIMIT_PLUS2

Check the axis-specific NC/PLC interface signals for selecting the 2nd software limit switch
- DB31, ... DBX12.2 and .3

Check currently active work offsets of the current frame.
Also check external work offsets, overlaid movements ($AA_OFF), DRF and transformation components.

Programm 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

Explanation: The alarm is triggered during the preparation of the displayed approach or residual block for repositioning on the contour (REPOS).
Cause:
The programmed path violates the active software limit switch in the displayed traversing direction for the displayed axis.
Note:
Parameter 4 contains the descriptions of parameters 5, 6 and 7 separated by the character "|".
- 5 = Block number, label
- 6 = Number + string (+/-)
7 = Distance-to-go
Alarm 10723 is displayed for MD11411 $MN_ENABLE_ALARM_MASK, bit11 == 1 instead of alarm 10721.
Alarm 10723 offers an expanded diagnostics option for the software limit switch violation.
Condition:
Alarm file ALUN* is present in the HMI.
See also:
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 a user ASUB or system ASUB. See also cross reference from ASUBs.
Please inform the authorised personnel/service department.
Check the parameterisation of the software limit switch:
- MD36100 $MA_POS_LIMIT_MINUS
- MD36120 $MA_POS_LIMIT_MINUS2
- MD36110 $MA_POS_LIMIT_PLUS
- MD36130 $MA_POS_LIMIT_PLUS2
Check the axis-specific NC/PLC interface signals for selecting the 2nd software limit switch
- DB31, ... DBX12.2 and .3
Check the active work offsets of the current frame.
Check the external work offsets, overlaid movements ($AA_OFF), DRF and transformation components.

Programm 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 -)

Explanation: 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

Programm 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 -)

Explanation: 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.

Programm 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

Explanation: This alarm was triggered during the preparation of the displayed block.
Cause:
The programmed path violates the active working area limitation for the displayed axis.
Note:
Parameter 4 contains the descriptions of parameters 5 and 6 separated by the character "|".
- 5 = Block number, label
- 6 = Distance-to-go
Alarm 10732 is displayed for MD11411 $MN_ENABLE_ALARM_MASK, bit11 == 1 instead of alarm 10730.
Alarm 10732 offers an expanded diagnostics option for the working area limitation violation.
Condition:
Alarm file ALUN* is present in the HMI.
See also:
Diagnostics Manual for alarm 10730

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

Programm 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

Explanation: The alarm was triggered during the preparation of the displayed approach or residual blocks for repositioning on the contour (REPOS)
Cause:
The programmed path violates the active working area limitation for the displayed axis.
Note:
Parameter 4 contains the descriptions of parameters 5 and 6 separated by the character "|".
- 5 = Block number, label
- 6 = Distance-to-go
Alarm 10733 is displayed for MD11411 $MN_ENABLE_ALARM_MASK, bit11 == 1 instead of alarm 10731.
Alarm 10733 offers an expanded diagnostics option for the working area limitation violation.
Condition:
Alarm file ALUN* is present in the HMI.
See also:
Diagnostics Manual for alarm 10731

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.

Programm 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

Explanation: This alarm was triggered during the preparation of the displayed block.
Cause:
The programmed path of the displayed axis violates the coordinate system-specific working area limitation.
Note:
Parameter 4 contains the descriptions of parameters 5 and 6 separated by the character "|".
- 5 = Block number, label
- 6 = Distance-to-go
Condition:
Alarm file ALUN* is present 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 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.

Programm 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

Explanation: This alarm was triggered during the preparation of the displayed approach or residual block for repositioning on the contour (REPOS).
Cause:
The programmed path violates the coordination system-specific working field limitation of the displayed axis.
Note:
Parameter 4 contains the descriptions of parameters 5 and 6 separated by the character "|".
- 5 = Block number, label
- 6 = Distance-to-go
Condition:
The alarm file ALUN* is present 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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 TRC with G41/G42, program a tool number under the address T...

Programm 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

Explanation: 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 change time.

Programm 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

Explanation: 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.
Simplify 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).
- Increase the number of reviewed blocks for collision monitoring (MD20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 present 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

Programm 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

Explanation: 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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10757 [Channel %1:] Block %2 Impermissible change of orientation while tool radius compensation is active

Parameters: %1 = Channel number

%2 = Block number, label

Explanation: An orientation change has been programmed which is not permissible with the active tool radius compensation type (G code of group 22, e.g. CUT2D or CUT2DF). As a rule, changes to the tool orientation are only permissible and useful if 3D tool radius compensation is active.

An (impermissible) orientation change can also be triggered by a change of machining plane (G17 - G19).

Reaction: Correction block is reorganized.

Local alarm reaction.

Interface signals are set.

Alarm display.

NC Stop on alarm at block end.

Remedy: Activate a G code from group 22 (e.g. CUT3DC), in which the programmed orientation change is permissible.

Execute the program with constant tool orientation.

If a plane change is necessary, first deactivate the tool radius compensation, and reactivate after the plane change.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10785 [Channel %1:] Block %2 Impermissible tool for TRC with difference tool

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

Explanation: Neither a turning nor a grinding tool nor any other tool with a relevant cutting edge position is permitted during the activation of the tool radius compensation with a difference tool.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 speed under address S in [m/min]. It is derived from the current spindle speed.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 acknowledgment 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: 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].

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: This alarm is output if, when inserting chamfers or radii, at least one of the relevant blocks is so short that the contour element to be inserted must be reduced from its originally programmed value. The alarm occurs only if bit 4 is set in MD11411 $MN_ENABLE_ALARM_MASK. Otherwise, the chamfer or rounding 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.

Programm continuation: Clear alarm with the Delete key or NC START.

10890 [Channel %1:] Block %2 overflow of local block buffer when calculating splines

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

Explanation: 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.

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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10891 [Channel %1:] Block %2 multiplicity of node is greater than its order

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

Explanation: In the B spline the distance between nodes PL (node = point on spline at which 2 polynomials meet) has been programmed with zero too often in succession (i.e. the "multiplicity" of a node is too great).
In the quadratic B spline the node distance may not be specified more than twice with 0 in succession, and in the cubic B spline not more than 3 times.

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

Remedy: Program the node distance PL = 0 in succession no more than the degree of the B spline used.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10910 [Channel %1:] Block %2 irregular velocity waveform of one path axis

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

Explanation: When the path axis waveforms were analyzed during block preparation, a large local deviation relative to the path velocity was detected in the velocity waveform of one or more path axes.
Such a situation can have the following causes:
- The path runs close to singular positions of the machine kinematics.
- The programmed contour characteristic is very uneven.
- The FGROUP definition is unfavorable relative to the contour.
- The setting MD28530 $MC_MM_PATH_VELO_SEGMENTS=0 is inadequate for curvature changes occurring within one block. This problem occurs more frequently with G643, G644 and COMPCAD.
- A kinematic transformation has been implemented with insufficient numerical accuracy.
The path velocity is generally reduced substantially in order to avoid axis overloads safely. An apparent machine standstill may occur. Severe axis movements occur suddenly as soon as the singular position is reached.

Reaction: Alarm display.
Warning display.

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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10911 [Channel %1:] Block %2 transformation prohibits to traverse the pole

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

Explanation: The given curve passes through the pole of the transformation.

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

Remedy: Modify part program.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

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

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

Reaction: Alarm display.

Remedy: No action is usually necessary. The velocity control operates more effectively, however, if the part program is modified.
- If a transformation pole is circumnavigated several times by the curve, it helps to split up the block into smaller parts.
- If a positioning axis is the cause, you should check whether the axis can be traversed as a path axis. The Look Ahead function remains deactivated until preprocessing can be based on defined conditions again (e.g. as a result of change from JOG->AUTO, tool or tool edge change).

Programm continuation: Clear alarm with the Delete key or NC START.

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

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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

10914 [Channel %1:] Block %2: movement not possible while transformation active.

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

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

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

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

Programm continuation: Clear alarm with the RESET key. Restart part program

10915 [Channel %1:] Block %2 Preparation problem in LookAhead (Identifier %3, Details %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Error code
%4 = Error details

Explanation: The NCK was incorrectly parameterized (under certain circumstances, the parameterized memory is not sufficient), which is why LookAhead can no longer be operated in the expansion mode.

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

Remedy: Change parameterization (increase LookAhead memory and/or IPO buffer, change tolerances). Use standard LookAhead.
Contact Siemens if necessary.

Programm continuation: Clear alarm with the RESET key. Restart part program

10916 [Channel %1:] Block %2 Preparation problem in LookAhead (Identifier %3, Details %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Error code
%4 = Error details

Explanation: The NCK was incorrectly parameterized (under certain circumstances the parameterized memory is not sufficient), this is the reason that the generated profile is not as smooth and uniform as it could be.

Reaction: Local alarm reaction.
Alarm display.
Warning display.

Remedy: Change the parameterization (increase the LookAhead memory and/or IPO buffer, change tolerances).
1010: Increase IPO buffer to at least 50 blocks, or maximum number of blocks in the braking ramp times 2.

Programm continuation: Clear alarm with the Delete key or NC START.

10917 [Channel %1:] Block %2 Warning of COMPSURF function (code %3, details %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Error code
%4 = Error details

Explanation: COMPSURF was only able to work with restrictions.
The warning is only displayed if MD11411 $MN_ENABLE_ALARM_MASK bit1 is set.

Reaction: No alarm reaction.
Warning display.

Remedy: 1: Modify parameterization (smaller tolerance, larger block buffer).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: 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).
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm continuation: Clear alarm with the RESET key. Restart part program

10950 [Channel %1:] Calculation of arc length function too inaccurate

Parameters: %1 = Channel number
Explanation: The calculation of the arc length function could not be performed to the required accuracy.

Reaction: Alarm display.
Warning display.

Remedy: The calculation of the arc length function could not be performed to the required accuracy during active polynomial interpolation. Either increase MD20262 $MC_SPLINE_FEED_PRECISION or reserve more memory for the representation of the arc length polynomials. MD28540 $MC_MM_ARCLENGTH_SEGMENTS defines how many polynomial segments can be used per block in order to approximate the arc length function.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10960 [Channel %1:] Block %2 COMPCURV/COMPCAD and radius compensation can-notbe used simultaneously

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

Explanation: Compressor types COMPCURV and COMPCAD cannot be used in combination with tool radius compensation. Only compressor type COMPON can be activated while 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10961 [Channel %1:] Block %2 maximum cubic polynomials are allowed on active radius compensation.

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

Explanation: With active radius compensation, only up to cubic polynomials are permissible for the geometry axes. In this case no 4th or 5th degree polynomials can 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

10963 [Channel %1:] Block %2 COMPSURF is unable to smooth adequately

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

Explanation: Smoothing by COMPSURF is not optimal because limited by internal memory sizes.
The alarm only occurs if MD11400 $MN_TRACE_SELECT bit10 is set; otherwise only warning 10917 is output.

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

Remedy: Reduce tolerance (CTOL, OTOL, ATOL) in the part program.

Programm continuation: Clear alarm with the RESET key. Restart part program

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

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

Explanation: 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).

Programm 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

Explanation: The number of times each address type may occur in an 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 block-by-block type!).

Programm 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

Explanation: 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 block-by-block address modifications only for permissible addresses, in accordance with the Programming Manual.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12030 [Channel %1:] Block %2 invalid parameter or data type in %3

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Source string

Explanation: In polynomial interpolation, polynomials must not be greater than the 3rd degree (refer to Programming Guide).

f(p) = a0 + a1 p + a2 p2 + a3 p3

The coefficients a0 (the starting points) are identical to the end points of the preceding block and need not be programmed. In the polynomial block, a maximum of 3 coefficients per axis is therefore allowed (a1, a2, a3).

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.

Programm 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 group

Explanation: 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.

Programm 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 group

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: The grammar in this block is incorrect at the text position indicated. The precise reason for this error cannot be specified in more detail because there are too many possibilities for error.
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
Example 3:
N13 R1=5
N15 R1=10 M=R1 ; value assignments must stand alone in the block, there must not be any other commands, such as auxiliary function outputs or traversing.
The cause may also be the use of Unicode characters. This is indicated by the representation of characters not included in the ASCII character set in the alarm text.
Whether a character not included in the ASCII character set is transformed into a space or alarm 12080 is output can be set using bit 3 of MD 10280 $MN_PROG_FUNCTION_MASK.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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)

Explanation: 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.

Programm 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

Explanation: 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)

Programm 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

Explanation: 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).

Programm 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

Explanation: 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 speed [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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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 group

Explanation: 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.

Programm 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

Explanation: 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

Programm 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 group

Explanation:
- 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: Only the R-variables 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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].

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 variables, or by grouping together parameters of the same type to form an array and transfer them in this form.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 variables), 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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 PROGRAM CORRECT. The cursor jumps to the incorrect block. Then delete the invalid program attribute.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12490 [Channel %1:] Block %2 access permission level %3 was not set

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

Explanation: The desired access authorization was not set. The desired protection level is either beyond the permitted value range or changing the protection level is not allowed.
The protection level may be changed only if:
1. The current protection level is equal to or higher than the level originally defined, and
2. The new protection level will be below the level originally defined.
The alarm also occurs if the access authorization is to be changed for a user data block that is not available.
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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12550 [Channel %1:] Block %2 name %3 is not known or not defined

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

Explanation: The identifier displayed is not known or was not defined before use.
A definable identifier can be:
A macro, GUD, LUD, program name or a program parameter

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
- See also MD10711 $MN_NC_LANGUAGE_CONFIGURATION.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12551 [Channel %1:] Block %2 motion synchronized action: %3 function not available

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID

Explanation: Motion synchronized action: this function is not available for this system.

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

Remedy: - Correct the name used (write error)
- Use a higher-level software system for reduced functions
- Check the definitions of variables, subroutines and macros
- Declare a subroutine with EXTERN; load the subroutine to SPF-Dir
- Check the interface definition of the subroutine

Programm 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.

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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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)

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: The identifier is not available 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 faulty 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: +/-(2.2*10e-308 ... 1.8*10e308)
- INT: Property: Integers with signs, value range: -2147483648 ... +2147483647
- BOOL: Property: Truth value FALSE, TRUE, value range: 0 or 1
- CHAR: Property: 1 ASCII character, value range: 0 ... 255
- STRING: Property: Character string (max. length depends on the variable), value range: 0 ... 255
- AXIS: Property: Axis addresses, value range: Channel identifier
- FRAME: Property: Geometric information, value range: ---

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: $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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: - 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: Global user data cannot be created because the user data block for this global user data is not available.
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.

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.

Programm 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

Explanation: 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").

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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

Explanation: 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 variables are programmed with R1.

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

Remedy: Modify part program.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12661 [Channel %1:] Block %2 technology cycle %3: no further subprogram call possible

Parameters: %1 = Channel number
%2 = Block number
%3 = Name of the technology cycle call

Explanation: In a technology cycle it is not possible to call a subroutine or another technology cycle.

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

Remedy: Modify part program.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12712 [Channel %1:] Block %2 external language mode is not active

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

Explanation: It is not possible to switch to external language mode.
Before external language mode can be used, it has to be configured (see MD18800 $MN_MM_EXTERN_LANGUAGE).

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

Remedy: Modify part program.
Configure external language mode.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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].

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

12770 [Channel %1:] Block %2 Conversion not possible. Error code %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Cause of the error

Explanation: When converting a part program using the G code converter function, an error was identified.
- Error cause: 1 = no Jobshop program. Conversion is only permissible for Jobshop -programs
- 2 = an LUD call by reference variable as transfer parameter at a predefined function or a cycle that cannot be converted was transferred
- 3 = G code conversion is not possible in the AUTO mode
- 4 = an attempt was made to make a conversion in the ISO mode, (check G291, $MC_GCODE_RESET_VALUES[46])
- 5 = for the output into the trace program, sufficient memory was no longer available, therefore the trace program was deleted.

Reaction: Alarm display.
No alarm reaction.

Remedy: Do not activate G code converter in the ISO mode

Programm continuation: Clear alarm with the Delete key or NC START.

12780 [Channel %1:] Block %2 instruction %3 not allowed for precompiling the program

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

Explanation: Not all instructions are available in compile mode.
The invalid language command is shown in the alarm text.

Reaction: Alarm display.

Remedy: Correct part program.
With external instruction:
- Store the subprogram in cycle directory _N_CUS_DIR, _N_CMA_DIR or _N_CST_DIR
- If bit 0 = 1 (default value) in MD10700 $MN_PREPROCESSING_LEVEL, then the EXTERNAL instruction is no longer required

Programm continuation: Clear alarm with the Delete key or NC START.

14000 [Channel %1:] Block %2 illegal end of file

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

Explanation: Alarm 14000 is output in the following situations:
- Part program was not terminated with M30, M02 or M17.
- Executing from external: Download was canceled (e.g. because HMI was switched off).
- Executing from external: REPEAT loop does not lie completely within the reload buffer

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

Remedy:
- End part program with M30, M02 or M17 and start part program.
- Executing from external: If the download for the selected program was canceled,
  the default program _N_MPF0 is automatically selected with RESET.
  The selection of the user program must be repeated after that.
- Executing from external with REPEAT command: Replace REPEAT loop by EXTCALL call.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

14004 [Channel %1:] Program %2 cannot be started because of a channel-specific start disable

Parameters: %1 = Channel number
%2 = (path with program name)

Explanation: The selected program %2 in channel%1 cannot be executed because the channel-specific start disable is set for this channel.
Background: When ShopMill or ShopTurn programs are selected or changed, the HMI performs a consistency check of the parameters that have been entered. During this time, the processing of the selected NC program is inhibited by the HMI setting what is known as the "channel-specific start disable". If an NC start is now rejected because of a set channel-specific start disable, then alarm 14004 is output as a function of MD 11411 $MN_ENABLE_ALARM_MASK bit 15. The alarm is also output in the event of a block search, but in this case it is not dependent on MD 11411 $MN_ENABLE_ALARM_MASK bit 15.

Reaction: Alarm display.

Remedy: Repeat NC start

Programm continuation: Clear alarm with the Delete key or NC START.

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

Explanation: Program %3 cannot be executed, because the program-specific start disable has been set for this file.
Background: When an editing operation for a ShopMill or ShopTurn program has been completed, the HMI performs a consistency check of the parameters that have been entered. During this time, CNC program processing is inhibited by setting what is known as the "program-specific start disable" by the HMI. If NC start is pressed during the test, the start is not executed, and alarm 14005 is output as a function of MD 11411 $MN_ENABLE_ALARM_MASK bit 6.
Alarm 14005 is also output if the part program processing encounters a subroutine for which the file attribute "start disable" is set.
After the test step has been completed, the program processing can continue by pressing NC start.

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

Remedy: Repeat NC start

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

14007 [Channel %1:] Block %2 Program %3 is being edited

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

Explanation: Program %3 cannot be executed because it has been disabled by another application, for example the HMI Editor.
Background: Program %3 is on an external data carrier (CF card, network drive, USB device), and should be executed from there in EES mode (Execution from External Storage). But the program cannot be processed because it has been opened for writing by another application, for example the HMI Editor, and the WRITE lock has been set for this file.

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

Remedy: Close the application that set the WRITE lock, that is for example the HMI Editor, and continue the program processing with NC start.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14008 [Channel %1:] Block %2 WRITE command writes in the temporary memory area in /_N_EXT_DIR

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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: A subroutine call was canceled because the called subroutine could not be opened.
The subroutine call can be executed via
- subroutine identifier
- 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 part program memory
- 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 directory (max. of 24 characters). programmName: Name of subroutine (max. of 24 characters)
- A reload buffer for executing from external was called as subroutine.

Note: Unknown identifiers (string) found in the part 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 part 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.

Programm 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

Explanation: The maximum nesting depth of 16 program levels has been exceeded.
Subroutines can be called from the main program, and these in turn may have a nesting depth of 15.
For interrupt routines, two additional program levels can be used. This means that the total number of program levels is increased to 18.
The program levels are jointly used by user programs and Siemens cycles and/or Siemens applications such as ShopMill and ShopTurn.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: The selected part program is not in the NCK memory or the execution right for the part program has a higher level than the current access right.
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 one of the specified definition files 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.

Programm 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

Explanation: 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)

Programm 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

Explanation: 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
- The end of part program is programmed
- An M98 subprogram call is active (only in external language mode)
- A T function replacement is programmed with D function programming in the same part program line with active TLC (G43/G44) in the ISO2 system.
- A T function replacement is programmed, configured at the end of block, and a subprogram call, for example by a modal cycle call.

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.
If a T function replacement is configured with execution at end of block and a subprogram call is programmed in the same block, the T function replacement must be executed at start of block. This means that MD10719 $MN_T_NO_FCT_CYCLE_MODE bit 1 = 1 must be set.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14018 [Channel %1:] Block %2 part 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

Explanation: To part 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 part program. Please see the Siemens Programming Guide or OEM documentation for the language commands permissible for the relevant system configuration.

Programm 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

Explanation: - 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.

Programm 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

Explanation: - 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.

Programm 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

Explanation: - 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.
Programm 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
Explanation: 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.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14024 [Channel %1:] Block %2 No access authorization for initialization of %3[%4].

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Variable to be reset
%4 = Index of the variable to be reset
Explanation: When the function DELOBJ is called, an attempt is made to reset a variable to its default value. The access authorization is not sufficient for this purpose.
When this alarm occurs, no data are changed, not even that for which the access authorization would be sufficient.
This alarm can only occur when the current value of the system variable, to which the attempt to write led to the alarm, is not equal to its initial setting value.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Increase access authorization.
Programm 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
Explanation: 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.
Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 MD21010 $MC_CIRCLE_ERROR_CONST divided by MD21010 $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 MD21010 $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.

Programm 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

Explanation: 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.

Programm 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

Explanation: In the circle programming, an inadmissible number of full revolutions has been specified.

The number of revolutions must not be negative and must not be greater than 1000000.

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

Remedy: Modify part program.

Programm 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

Explanation: 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.

Programm 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

Explanation: - 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.

Programm 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

Explanation: 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 block-by-block 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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 ISOOPEN, 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.

Programm 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

Explanation: 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?

Programm 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

Explanation: 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?

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: One of the following programming errors has occurred:

1. The keyword WAITP(x) "Wait with the block change until the specified positioning axes has reached its end point", was used for an axis that is not a positioning axis.

2. G74 "Reference point approach from the program" was programmed for a spindle. (Only axis addresses are permissible.)

3. The keyword POS/POSA was used for a spindle. (For spindle positioning, keywords SPOS and SPOSA should be programmed.)

4. If the alarm occurs for the function "Thread tapping without compensating chuck " (G331) the following reasons are conceivable:

- The master spindle is not in position-controlled operation.
- Incorrect master spindle
- Master spindle without encoder

5. An axis name that is no longer available has been programmed, e.g. when using axial variables as index. Or it was programmed as index NO_AXIS.

6. If 14092 is output as note for alarm 20140 motion synchronized action: traverse the command axis, then the the following causes are possible:

- The axis is presently being traversed by the NC program.
- An overlaid movement is active for the axis.
- The axis is active as slave axis of a coupling.
- An interpolation compensation such as a temperature compensation is active for the axis.

7. PRESETON/ PRESETONS were programmed for an unknown channel 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14093 [Channel %1:] Block %2 path interval <= 0 with polynomial interpolation

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

Explanation: In the polynomial interpolation POLY, a negative value or 0 has been programmed under the keyword for the polynomial length PL=...

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 value given in PL = ...

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14094 [Channel %1:] Block %2 polynomial degree greater than 3 programmed for polynomial interpolation

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

Explanation: The polynomial degree in the polynomial interpolation is based on the number of programmed coefficients for an axis. The maximum possible polynomial degree is 3, i.e. the axes are according to the function:

f(p) = a0 + a1 p + a2 p2 + a3 p3

The coefficient a0 is the actual position at the start of interpolation and is not programmed!

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

Remedy: Reduce the number of coefficients. The polynomial block may have a form no greater than the following:
N1 POLY PO[X]=(1.11, 2.22, 3.33) PO[Y]=(1.11, 2.22, 3.33)
N1 PO[n]=... PL=44
n ... n ... axis identifier, max. 8 path axes per block

Programm 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

Explanation: 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.

Programm 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

Explanation: 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: +/-(2.2*10e-308 ... 1.8*10e308)
- INT: Property: Integers with signs, value range: -2147483648 ... +2147483647
- BOOL: Property: Truth value FALSE, TRUE, value range: 0 or 1
- CHAR: Property: 1 ASCII character, value range: 0-255
- STRING: Property: Character string (max. length depends on the variable), value range: 0 ... 255
- AXIS: Property: Axis addresses, value range: Channel identifier
- FRAME: Property: Geometric information, value range: ---

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.

Programm 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

Explanation: 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").

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

14102 [Channel %1:] Block %2 polynomial degree greater than 5 programmed for orientation vector angle

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

Explanation: During polynomial interpolation for the orientation vector, a polynomial degree larger than 5 has been programmed.

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

Remedy: Modify part program.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14103 [Channel %1:] Block %2 error %3 when calling function CORRTRAFO.

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

Explanation: An error occurred when the CORRTRAFO function was called. The error number specifies the cause of the error.
Error numbers:
1: No transformation active.
2: The current active transformation is not an orientation transformation.
3: The active orientation transformation was not defined with kinematic chains.
10: The call parameter _CORR_INDEX is negative.
11: The call parameter _CORR_MODE is negative.
12: Invalid reference to the section of a partial chain (1st position of _CORR_INDEX). The value must not be greater than the number of orientation axes in the partial chain.
13: Invalid reference to the orientation axis of a partial chain (1st position of _CORR_INDEX). The value must be less than the number of orientiation axes in the partial chain.
14: Invalid reference to a partial chain (10th position of _CORR_INDEX). Only the values 0 and 1 are permissible (reference to part or tool chain). This error number also occurs if the partial chain to which _CORR_INDEX refers, does not exist.
15: In the section referred to with the parameter _CORR_INDEX, no correction element is defined($NT_CORR_ELEM_P or $NT_CORR_ELEM_T).
20: Invalid correction mode (1st position of _CORR_MODE). Only the values 0, 1, 8, and 9 are permissible.
21: Invalid correction mode (10th and / or 100th position of _CORR_MODE). When reading a system variable, the 100th position must always be zero. When reading or writing an axis direction, only the one's position is permitted to be not equal to zero.
30: The 100th position of _CORR_MODE is invalid. Only the values 0 and 1 are permissible.
31: The 1000th position of _CORR_MODE is invalid. Only the values 0 and 1 are permissible.
40: The direction vector to be taken over as the axis direction is the zero vector.
41: In the correction of an offset vector, the deviation from the current value is greater than the maximum value set in setting data 41610 $SN_CORR_TRAFO_LIN_MAX in at least one coordinate.
42: In the correction of a direction vector, the angle deviation from the current direction is greater than the maximum value set in setting data 41611 $SN_CORR_TRAFO_DIR_MAX.
43: The attempt to write a system variable was rejected due to missing write authorization.

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

Remedy: Change function call.

Programm continuation: Clear alarm with the RESET key. Restart part program

14109 [Channel %1:] Block %2 simultaneous linear and rotary axis movement with static orientation transformation.

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

Explanation: The interpolation type CP (G group code 49) is not permitted with an active static orientation transformation if linear and rotary axes have to interpolate simultaneously.

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

Remedy: Change the NC program:
Activate dynamic orientation transformation.
Change the G code for group 49.
Execute linear and rotary axis movement successively rather than simultaneously.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

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

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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

14144 [Channel %1:] Block %2 PTP movement not allowed, error code %3.

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

Explanation: The desired PTP movement is not possible. The cause of the error is described in more detail by the error code.
Error code:
- 1. A PTP interpolation is not possible with the currently active transformation.
- 2. PTP interpolation and tool radius compensation must not be active simultaneously.
- 3. The PTP interpolation types PTPWOC and PTPWOC2 are only permissible with orientation transformations.
- 4. PTP interpolation and COMPSURF must not be active simultaneously.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Modify the program.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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=...).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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=...).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14290 [Channel %1:] Block %2 polynomial degree greater than 5 programmed for polynomial interpolation

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

Explanation: A polynomial degree greater than five was programmed for the polynomial interpolation. You can only program polynomials up to the 5th degree.

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

Remedy: Modify part program.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14401 [Channel %1:] Block %2 transformation not available. Error code %3.

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

Explanation: The required transformation is not available. The cause of the error is explained more closely by the error code.
1: No transformations are available.
2: No transformations of the type "inclined axis" are available.
3: No orientation transformations are available.
4: No transmit transformations are available.
5: No tracyl transformations are available.
6: No chained transformations are available.
7: No OEM transformations are available.
8: No OEM orientation transformations are available.
22: The required transformation of the type "inclined axis" was not found.
23: The required orientation transformation was not found.
24: The required transmit transformation was not found.
25: The required tracyl transformation was not found.
26: The required chained transformation was not found.
40: An attempt was made to activate a transformation defined by kinematic chains. However, no such transformations have been enabled (MD18866 $MN_MM_NUM_KIN_TRAFOS is zero).
41: No transformation name (first call parameter) was specified for the transformation call TRAFOON.
42: An attempt was made to activate a transformation defined by kinematic chains. However, no such transformation with the specified name was found.
43: An attempt was made to activate a transformation defined by kinematic chains whose name had been changed since the last activation. Changed transformation data does not become active until after a NEWCONF or RESET.
44: An attempt was made to activate a transformation defined by kinematic chains. But the stated name is empty (zero string).
45: An attempt was made to activate a transformation defined by kinematic chains. However, the system variables $NT_NAME[..] are all empty.
46: An attempt was made to activate a transformation defined by kinematic chains. The stated transformation name was found more than once in the array of system variables $NT_NAME.
52: An attempt was made to activate a RESET transformation defined by kinematic chains (MD20142 $MC_TRAFO_RESET_NAME). However, no transformation was found with the stated name.
55: An attempt was made to activate a RESET transformation defined by kinematic chains (MD20142 $MC_TRAFO_RESET_NAME). However, the system variables $NT_NAME[..] are all empty.
56: An attempt was made to activate a RESET transformation defined by kinematic chains (MD20142 $MC_TRAFO_RESET_NAME). However, the stated transformation name was found more than once in the array of system variables $NT_NAME.

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

Remedy: Error codes 1 to 6 indicate that the control does not contain the requisite software. The transformation cannot even be activated by setting machine or option data.
For the remaining error codes:
Modify part program; program defined transformations only.
Check MD24... $MC_TRAFO_TYPE_... (assigns the transformation to the part program operation).

Programm continuation: Clear alarm with the RESET key. Restart part program

14402 [Channel %1:] Block %2 spline active at transformation change

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

Explanation: A change of transformation is not allowed in a spline curve section. A series of spline blocks must be concluded.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14403 [Channel %1:] Block %2 preprocessing and main run might not be synchronized

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Positioning axis runs cannot be accurately calculated beforehand. Consequently, the position in the Machine (MCS) is not known exactly. It might therefore be possible that a change in the multiple significance of the transformation has been performed in the main run although no provision was made for this in the preprocessing run.
Reaction: Alarm display.
Remedy: Modify part program. Synchronize preprocessing run and main run.
Programm continuation: Clear alarm with the Delete key or NC START.

14404 [Channel %1:] Block %2 illegal parameterization of transformation

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Error has occurred when selecting transformation.
Possible causes of error:
- An axis traversed by the transformation has not been enabled:
- is being used by another channel (-> enable)
- is in spindle mode (-> enable with SPOS)
- is in POSA mode (-> enable with WAITP)
- is concurrent Pos axis (-> enable with WAITP)
- Parameterization via machine data has an error
- Axis or geometry axis assignment to the transformation has an error,
- Machine data has an error (-> modify machine data, restart)
Please note: Any axes not enabled might be signaled via alarm 14092 or alarm 1011 instead of alarm 14404.
Transformation-dependent error causes can be in: TRAORI: -
TRANSMIT:
- The current machine axis position is unsuitable for selection (e.g. selection in the pole) (-> change position slightly).
- Parameterization via machine data has an error.
- Special requirement with respect to the machine axis has not been fulfilled (e.g. rotary axis is not a modulo axis) (-> modify machine data, restart).
TRACYL:
The programmed parameter is not allowed when transformation is selected.
TRAANG:
- The programmed parameter is not allowed when transformation is selected.
- Parameterization via machine data has an error.
- Parameter is faulty (e.g. TRAANG: unfavorable angle value (-> modify machine data, restart)
Persistent transformation:
- Machine data for persistent transformation are wrong (-> consider dependencies, change machine data, restart)
Only with active "OEM transformation" compile cycle:
The axes included in the transformation must be referenced.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Modify part program or machine data.
Only with active "OEM transformation" compile cycle:
Reference the axes included in the transformation before selecting transformation.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14405 [Channel %1:] Block %2 Transformation call refers to $NT_NAME[%4] = %3.

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

Explanation: The transformation call is ambiguous. This means the transformation designated in the alarm text and defined by kinematic chains is parameterized (system variable $NT_TRAFO_INDEX) so that it can be activated with the conventional transformation call in the error line.
The associated conventional transformation is also defined at the same time.
Only one of the two transformations may be present.

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

Remedy: Remove one of the two transformations involved.

Programm continuation: Clear alarm with the RESET key. Restart part program

14406 [Channel %1:] Block %2 Parameter error %3 on calling a transformation.

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

Explanation: An error was detected during the attempt to activate a transformation. The cause of the error is indicated more precisely by the following error code.
- 80. The geometry axes are (almost) linearly dependent, this means that two geometry axes are almost parallel or three geometry axes lie almost in one plane.
- 120. During the redirection of a TRAANG transformation to a transformation defined by kinematic chains, a parameter was stated as an angle that was not the same as the angle defined by the kinematic chains.
- 121. During the redirection of a TRAANG transformation to a transformation defined by kinematic chains, a parameter was stated as an angle, although the angle of the inclined axis was not defined. This case occurs if there is more than one inclined axis or if the inclined axis does not lie in a main plane.
- 150. The call of a TRACYL transformation did not state a value for the reference or work diameter.
- 200. The transformation can only be activated in a transformation chain, and not as a single transformation.

****

The following error codes are only output with OEM transformations if an error is detected during the selection of the transformation:
- 1000. The basic direction of the tool orientation must be parallel to one of the coordinate axes X, Y or Z.
- 1001. The basic direction of the tool orientation must not point in the X direction.
- 1002. The basic direction of the tool orientation must not point in the Y direction.
- 1003. The basic direction of the tool orientation must not point in the Z direction.
- 1004. The normal vector of the tool in the basic position must be parallel to one of the coordinate axes X, Y or Z.
- 1005. The normal vector of the tool in the basic position must not point in the X direction.
- 1006. The normal vector of the tool in the basic position must not point in the Y direction.
- 1007. The normal vector of the tool in the basic position must not point in the Z direction.

Note: The error codes 1000 to 1007 can only occur if $NC_IGNORE_TOOL_ORIENT = FALSE (default value), and the active tool contains orientations that are not permissible.
In this case, basic directions defined in the variables $NT_BASE_ORIENT and $NT_BASE_ORIENT_NORMAL do not become active

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Correct incorrect parameterisation.
Programm continuation: Clear alarm with the RESET key. Restart part program

14410 [Channel %1:] Block %2 spline active at geometry axis changeover

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: It is not allowed to change the assignment of geometry axes to channel axes in a spline curve definition.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Programm 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
Explanation: 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.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14412 [Channel %1:] Block %2 transformation active at geometry axis changeover

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: It is not permissible to change the assignment of geometry axes to channel axes when transformation is active.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 canceled 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.

Programm 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

Explanation: 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: - Cancel program with reset.
- Correct program on HMI or PC.
- Restart reloading (possibly with block search and interrupt location).

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

14620 [Channel %1:] Block %2 Error on opening program %3

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

Explanation: Program %3 cannot be executed because it has been locked by another application, e.g. HMI Editor.
Background: Program %3 is stored on an external data medium (CF Card, network drive, USB device), and should be executed from there in EES mode (execution from external storage). However, program %3 cannot be edited because it is locked by another application, e.g. HMI Editor, is open for writing and WRITE lock is set for this file.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Cancel program with Reset
- Close the application that set the WRITE lock, for example the HMI Editor, and continue program editing with NC Start.
Programm continuation: Clear alarm with the RESET key. Restart part program

14621 [Channel %1:] Block %2 Timeout on access to external program %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Program name
Explanation: Program %3 is stored on an external data medium (CF card, network drive, USB device). A timeout occurred when the program was accessed.
Possible cause of error: network disturbance.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Rectify network disturbances if applicable
Programm continuation: Clear alarm with the RESET key. Restart part program

14622 [Channel %1:] Access to file %2 disrupted by %3 %4

Parameters: %1 = Channel number
%2 = File which could not be accessed
%3 = Function with which it is accessed
%4 = Error message
Explanation: The program is stored on an external data medium (CF card, network drive, USB device). An error occurred when the program was accessed.
Possible cause of error: network disturbance.
Reaction: Alarm display.
Remedy: - Rectify network disturbances if applicable
Programm continuation: Clear alarm with the Delete key or NC START.

14623 The EES library does not exist

Explanation: The EES library does not exist; no part programs can be executed with EES.
Reaction: Channel not ready.
Alarm display.
Remedy: Load EES library
Programm continuation: Switch control OFF - ON.

14624 The EES library is incompatible.

Explanation: The EES library is incompatible; no part programs can be executed with EES.
Reaction: Channel not ready.
Alarm display.
Remedy: Replace EES library

Programm continuation: Switch control OFF - ON.

14625 [Channel %1:] Block %2 Problems in EES mode accessing file %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = File which could not be accessed

Explanation: The program is located on an external data medium (network drive, USB device). Problems occurred when accessing the program.
Possible cause or error: network fault.

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

Remedy: - if necessary, remedy network faults und resume program

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14650 [Channel %1:] Block %2 SETINT instruction with invalid ASUB input

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

Explanation: Asynchronous subroutines (ASUBs) are subroutines that are executed following a hardware input (interrupt routine started by a rapid NC input).
The NC 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 NC input of the SETINT statement with a value of not less than 1 or greater than 8.

Programm 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

Explanation: The NC 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 NC 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 NC input must be 1 or 2.

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

Remedy: Program the priority of the NC input with a value of not less than 1 or greater than 128.

Programm 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

Explanation: 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 if the program or operator action are repeated.
If you experience such a system error, please contact Technical Support.
www.siemens.com/sinumerik/help
Please supply the following information to ensure quick processing:
- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 areas 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

105: Error after WRITE lock on the selected program

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 area and an identifier of what is incorrect about the protection area. The system variables must be appropriately corrected.
If Parameter %3= 200 bis 203:
Increase MD28070 $MC_MM_NUM_BLOCKS_IN_PREP.

Programm continuation: Clear alarm with the RESET key. Restart part program

14711 [Channel %1:] Transformation selection not possible as axis %2 not available

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

Explanation: Based on the configuration of machine data MD20110 $MC_RESET_MODE_MASK and MD20140 $MC_TRAFO_RESET_VALUE, a transformation shall be selected by performing a reset or control ramp-up. However, this is not possible as axis %2 required for this is not available. Possible reason: The axis was occupied by another channel or the PLC.

Reaction: Interface signals are set.
Alarm display.

Remedy: - Use the GET command to get axis %2 in the channel in which the transformation is to be selected.
- Select the transformation by means of the part program command.

Programm continuation: Clear alarm with the RESET key. Restart part program

14712 [Channel %1:] Error selecting JOG Retract: error code %4 info %3

Parameters: %1 = Channel number
%2 = Is not used
%3 = Additional information
%4 = Error code

Explanation: An error occurred on the selection of JOG Retract, which is described in more detail by the error code (parameter%4):
List of error codes:
1: No retract data available
2: Selection is active during taper turning
3: Error preparing the initialization blocks. The additional info (parameter %3) states the initialization step in which the error occurred. The alarm output immediately before also refers to this initialization step:
100: Error synchronizing the preprocessing/main run
101: Error selecting the tool length compensation
102: Error selecting the transformation
103: Error generating the tool frame
104: Error generating the tapping block
105: Error replacing geometry axes
4: The position of the axis stated in the additional info does not have the status "synchronized" or "restored"
5: The axis stated in the additional info is already assigned in another channel by JOG Retract
6: The geometry axis stated by the selection of JOG Retract does not exist
7: MD 20110 $MC_RESET_MODE_MASK bit 0 is not set
8: Thread cutting is active. The thread direction cannot be explicitly assigned to a JOG axis

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

Remedy: The following conditions have to be fulfilled in order to select JOG Retract:
- A program execution with active tool offset has been canceled by Reset or PowerOff
- PLC signal DB3300.DBX4005.5 (Retract data available) or OPI variable retractState bit 1 is set
- JOG mode is active
- The channel in which JOG Retract is to be selected is in Reset state
- The function "Taper turning" must not be active when JOG Retract is selected
- Synchronized or restored axis positions must be present for the axes involved in the transformation
  If necessary, activate position restoration for incremental encoders (MD34210 $MA_ENC_REFP_STATE[]=3)
- MD 20110 $MC_RESET_MODE_MASK bit 0 must be set (default value)

In the case of an error, the alarm must be acknowledged with Reset. The selection can then be repeated, observing the above-mentioned conditions.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: In a group 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: Assignment: Illegal value.
The assignment to a variable or transfer parameter of a procedure or function was found to overshoot or undershoot the value range.

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

Remedy: Modify part program.
Comply with the value range of the variable or transfer parameter

Programm 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

Explanation: Programmed combination between action and type of motion synchronized 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 not allowed with WHENEVER, FROM and DO
- DELDTG and STOPREOF allowed only in block-by-block synchronized action with WHEN and EVERY
- PRESETON / PRESETONS may only be used together with WHEN or EVERY

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

Remedy: Modify part program.

Programm 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

Explanation: The synchronous variables $AA_LOAD, $AA_TORQUE, $AA_POWER and $AA_CURR are activated by the MD36730 $MA_DRIVE_SIGNAL_TRACKING. The system variable $VA_IS: Safe actual position is only available if the MD36901 $MA_SAFE_FUNCTION_ENABLE is set and the option $ON_NUM_SAFE_AXES is set to a sufficient size.

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

Remedy: Modify program or machine data.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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".

Programm 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

Explanation: The number of programmed PLC variables has exceeded the maximum permissible number.
For each write operation, one element is required for the following writing of PLC variables quickly one after the other.
If several write operations that are available as elements are to be executed, the block transport must be guaranteed (under certain circumstances, initiate preprocessing stop), or if available, increase MD28150 $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: Change the part program or where necessary, the machine data.

Programm 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

Explanation: 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 implicitely 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 implicitely 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14780 [Channel %1:] Block %2 option '%3<OPTNX>' is not set.

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Brief description of option

Explanation: A non-released option has been used in the block.
The stated or an equivalent option is required to execute the action.

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.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14781 [Channel %1:] Block %2 motion synchronized action: %3 Option '%4<OPTNX>' is not set

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Brief description of option
Explanation: Motion synchronized action: a non-enabled option was used.
The specified or an equivalent option is required to execute the action.
Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: Change motion synchronized action, retrofit option.
In this context, please compare the available option data and/or (if available) the license image of your control
Programm 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
Explanation: A non-active function is used in the block
Code Short description
1 Transformation
2 Tool H numbers
3 3D protection areas
4 Tool management, multitools
5 COMPSURF and MD28071 $MC_MM_NUM_SURF_LEVELS=0
6 TOFF (see OD19320 $ON_TECHNO_FUNCTION_MASK)
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: - Modify part program.
- Activate function.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14783 [Channel %1:] Block %2 coordinate system-specific working area limitation is not active

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: An attempt is made in the block to activate a group of the coordinate system-specific working area limitation.
However, this group is not set up (see MD28600 $MC_MM_NUM_WORKAREA_CS_GROUPS).
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
The NC program is stopped. The G code of the group WALCS01 - WALCS10 can be changed.

Remedy: - Modify part programm.
- Activate more coordinate system-specific working area limitations.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14784 [Channel %1:] Block %2 motion synchronous action: %3 function not possible

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID

Explanation: It is not possible to execute the function:

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

Remedy: Modify part program.

Programm continuation: Clear alarm with the RESET key. Restart part program

14790 [Channel %1:] Block %2 axis %3 programmed by PLC

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

Explanation: In the NC block, an axis has been programmed that is already being traversed by the PLC.

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

Remedy: - Modify part program, do not use this axis.
- Stop traversing motion of the axis by the PLC, modify part program (insert WAITP).

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14800 [Channel %1:] Block %2 programmed path velocity less or equal to zero

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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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.

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: The functions of constant grinding wheel peripheral 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

14860 [Channel %1:] Block %2 Selection of the tool cutting speed not allowed. Cause %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Cause of the error
Explanation: Selection of the cutting speed SVC is not permissible in the current state
Causes of the problem: the following function is active.
1: Constant cutting speed 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.
Programm 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
Explanation: Cutting speed 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.
Programm 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
Explanation: A cutting speed 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.

Programm 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

Explanation: The programmed value for the cutting speed 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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].

Programm 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

Explanation: 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.

Programm 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

Explanation: The INCH or METRIC instruction describes the system of measurement in which the data sets 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

15100 [Channel %1:] Block %2 REORG cancelation caused by log file overflow

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

Explanation: 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

Programm 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

Explanation: 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

Programm 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

Explanation: 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, part programs, R variables, 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.

Programm 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

Explanation: 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 diagnostics.

Programm continuation: Clear alarm with the RESET key. Restart part program

15150 [Channel %1:] Block %2 reload from external canceled

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

Explanation: Execution from external was canceled 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.
Examples:
- Definition of extremely long curve tables via execution from external.
- REPEAT command: REPEAT loop does not lie completely within the reload buffer.

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

Remedy: Insert machine function blocks into the part program.
- 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).
- REPEAT command: Replace REPEAT loop by EXTCALL call.

Programm continuation: Clear alarm with the RESET key. Restart part program

15160 [Channel %1:] Block %2 wrong preprocessing configuration, number of blocks %3, function ID %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of missing blocks for block preparation
%4 = ID of the function that identified the problem

Explanation: For the block preparation, the number of blocks specified in parameter %3 is also required.
Using parameter %4, for additional error diagnostics, the area in which the problem occurred can be identified based on the following list:
100 - 199: Interpreter
200 - 299: Tool radius compensation
300 - 399: Compile cycle
400 - 499: LookAhead
500 - 599: Spindle
600 - 699: Repositioning
700 - 999: Contour preparation
1000 - 1099: Nibbling
1100 - 1499: Tool orientation
1500 - 1599: Soft approach/retraction

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

Remedy: Please inform the authorized personnel/service department. Increase the block search configuration MD28070 $MC_MM_NUM_BLOCKS_IN_PREP by the number of blocks specified in parameter %3.

Programm continuation: Clear alarm with the RESET key. Restart part program

15165 [Channel %1:] Block %2 error when translating or interpreting ASUB %3

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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

15166 [Channel %1:] User system asub _N_ASUP_SPF not available

Parameters: %1 = Channel number

Explanation: By means of the MD11610 $MN_ASUP_EDITABLE the function "User-defined system ASUB" 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 ASUB is used.

Reaction: Interface signals are set.
Alarm display.

Remedy: Load the user-defined system ASUB in /_N_CUS_DIR/_N_ASUP_SPF or /_N_CMA_DIR/_N_ASUP_SPF laden.

Programm 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

Explanation: An error has occurred in compile mode. The (compiler) error message refers to the program specified here.

Reaction: Alarm display.

Remedy: Modify part program.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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").

Programm 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

Explanation: %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

Programm 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

Explanation: 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).

Programm 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

Explanation: 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).

Programm continuation: Clear alarm with the Delete key or NC START.

15189 [Channel %1:] Error executing SAFE.SPF

Parameters: %1 = Channel number

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 search target (typing error in the part program) and restart the search.

Programm continuation: Clear alarm with the RESET key. Restart part program

15360 [Channel %1:] Illegal target of block search (syntax error)

Parameters: %1 = Channel number

Explanation: The specified search target (block number, label or string) is not allowed in block search.

Reaction: Alarm display.

Remedy: Correct search target.

Programm continuation: Clear alarm with the Delete key or NC START.

15370 [Channel %1:] Target of block search not found

Parameters: %1 = Channel number

Explanation: 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.

Programm 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

Explanation: 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 target 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".

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

15700 [Channel %1:] Block %2 illegal cycle alarm number %3

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

Explanation: A SETAL command has been programmed with a cycle alarm number smaller than 60 000 or greater than 69 999.

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

Remedy: Program alarm number in the SETAL instruction in the correct range.

Programm 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

Explanation: A SETAL command has been programmed with a cycle alarm number smaller than 60 000 or greater than 69 999.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: LIFTFAST without interrupt routine (ASUB) 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 canceled with Reset.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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:
0x01 Axis is in another channel
0x02 Axis is in spindle mode (e.g. SPOS)
0x04 Axis is PLC axis
0x08 Axis is oscillating axis
0x10 Axis is neutral axis
0x20 Axis is coupled slave axis
0x40 Axis 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.

Programm 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

Explanation: 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 cancelled with alarm 16020.
In addition, the alarm is output in the following situations:
- Access to $AC_RETPOINT (repositioning point) outside an ASUB (e.g. in the main program)
- An axis to be repositioned was an oscillating axis with synchronous 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 an 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.
- Thread cutting (G33, G34, G35, G335, G336) was active in the interrupted block.

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

Remedy: Modify the part program if necessary.

Programm 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

Explanation: 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 canceled.

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
```

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: Programming of a speed is expected.

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

Remedy: Program speed with S[spindle number]=..

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16200 [Channel %1:] Block %2 spline and polynomial interpolation not available

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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16300 [Channel %1:] Block %2 denominator polynomial with zero places within parameter range not allowed

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

Explanation: The programmed denominator polynomial (with PL [ ] = ... , i.e. without specification of geometry axis) has a zero place within the defined parameter range (PL = ...). This means that the quotient of the numerator polynomial and the denominator polynomial is infinite or indeterminate.

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

Remedy: Modify the polynomial block so that there is no zero place within the polynomial length in the denominator polynomial.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16400 [Channel %1:] Block %2 positioning axis %3 cannot participate in spline

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

Explanation: An axis assigned to a spline grouping (n) with SPLINEPATH (n, AX1, AX2, ...) has been programmed as positioning axis with POS or POSA.

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

Remedy: Do not assign the positioning axis to the spline grouping.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16679 [Channel %1:] Block %2 Motion synchronized action: %3 Slave spindle/axis %4 not available

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name, spindle number

Explanation: A coupling was switched-in or switched-out, where the slave spindle/axis is presently not available. Possible causes are:
- The spindle/axis is active in the channel.
- The spindle/axis is active in the other channel.
- The spindle/axis was controlled from the PLC and has still not been enabled.

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

Remedy: Slave spindle/axis enabled with spindle/axis exchange or enabled from the PLC.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16735 [Channel %1:] Block %2 Incorrect geometry parameter (error %3)

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

Explanation: For G335/G336 (thread cutting with a convex thread) the pitch parameter could not be assigned to a velocity-defining axis. The error number specifies the precise cause:
Error 1 : The pitch parameter does not match the end points or the program circular plane.
Error 2 : A circular path angle of greater than 90 degrees was programmed.
Error 3 : The circular path has a helical component (helix).
Error 4 : The circular path exceeds one of the angles 45/135/225/315 degrees in the programming.
Error 5 : The circular path exceeds one of the angles 45/135/225/315 degrees after calculating in the frame.
Error 6 : The pitch was not programmed for the axis with the longest traversing path.

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

Remedy: - check circle programming or
- assign pitch parameter to the axis with the longest traversing distance.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm continuation: Clear alarm with the RESET key. Restart part program

16794 [Channel %1:] Block %2 coupling of axis/spindle %3 prohibits reference point approach

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

Explanation: The specified axis is a (gantry) slave axis and cannot therefore approach the reference point.

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

Remedy: Modify part program. Switch off coupling(s) of this axis before reference point approach or do not reference. A gantry slave axis cannot reference for itself.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

16800 [Channel %1:] Block %2 traverse instruction DC/CDC for axis %3 not allowed

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

Explanation: 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

Programm 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

Explanation: 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

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 canceled with Reset and repeat the operation.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm continuation: Clear alarm with the Delete key or NC START.

16906 [Channel %1:] Program control: action '%2<ALNX>' is canceled due to an alarm

Parameters: %1 = Channel number
%2 = Action number/action name
Explanation: The action was canceled due to an alarm.
Reaction: Alarm display.
Remedy: Eliminate the error and acknowledge the alarm. Then repeat the operation.
Programm 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
Explanation: This action may only be performed in Stop state.
Reaction: Alarm display.
Remedy: Check the program status and channel status.
Programm 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
Explanation: 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.
Programm 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
Explanation: A different operating mode must be activated for the activated function.
Reaction: Alarm display.
Remedy: Check operation and operating state.
Programm continuation: Clear alarm with the Delete key or NC START.

16911 [Channel %1:] Mode change is not allowed

Parameters: %1 = Channel number
Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: Incorrect mode change, e.g.: Auto -> MDIREF.

Reaction: Alarm display.

Remedy: Check operation or selected mode.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: An identical action is still being processed.
Reaction: Alarm display.
Remedy: Wait until the previous procedure has been terminated and then repeat the operation.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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!

Programm continuation: Clear alarm with the Delete key or NC START.

16924 [Channel %1:] Caution: program test modifies tool management data

Parameters: %1 = Channel number

Explanation: 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".

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: The language functions WAITMC, SETM, CLEARM and MSG must be programmed in separate NC blocks. To avoid velocity drops, these blocks are attached to the next NC block internally in the NCK (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.
Note: This does not apply to the programming of MSG with the additional parameter "1", in this case a separate block is generated, which interrupts the continuous-path mode.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 cancel the Reorg event handling! Reorg events are, e.g. cancelation 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: Cancel program with the RESET key.

Programm 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

Explanation: Reorg events are, e.g. cancelation 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 canceled with Reset.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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: Cancel program with the RESET key.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

16938 [Channel %1:] Action '%2<ALNX>' canceled due to active gear change

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

Explanation: Reorganization events are, among others, subprogram cancelation, 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 canceled with Reset and, if necessary, MD10192 $MN_GEAR_CHANGE_WAIT_TIME must be increased.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: Currently, the alarm occurs only in combination with the SERUPRO action. SERUPRO stands for search via program test.

SERUPRO is currently searching the 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

16943 [Channel %1:] Action '%2<ALNX>' not possible due to ASUB

Parameters: %1 = Channel number

%2 = Action number/action name

Explanation: 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 ASUB or a system ASUB has been triggered. User ASUBs are activated via FC-9 or via the fast inputs.

The following events lead to system ASUBS:

- 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

16946 [Channel %1:] Start via START is not allowed

Parameters: %1 = Channel ID

Explanation: 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).

Programm continuation: Clear alarm with the Delete key or NC START.

16947 [Channel %1:] Start via PLC is not allowed

Parameters: %1 = Channel ID

Explanation: 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).

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

16950 [Channel %1:] Search run with hold block

Parameters: %1 = Channel number

Explanation: 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

Programm continuation: Clear alarm with the Delete key or NC START.

16951 [Channel %1:] Search run in a protected program section.

Parameters: %1 = Channel number

Explanation: 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

Programm 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

Explanation: NCK is currently executing an ASUB in MDI mode. In this constellation, part program command "Start" is not allowed for another channel. Attention: If an ASUB 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

Programm 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

Explanation: 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

Programm 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

Explanation: 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

Programm continuation: Clear alarm with the RESET key. Restart part program

16955 [Channel %1:] Stop in stop delay area is delayed

Parameters: %1 = Channel ID

Explanation: 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

Programm 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)

Explanation: 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.
Programm continuation: Clear alarm with the Delete key or NC START.

16957 [Channel %1:] Stop-Delay area is suppressed

Parameters: %1 = Channel ID
Explanation: 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
Programm 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
Explanation: The function (2nd parameter) must not be activated during simulation search.
Reaction: Alarm display.
Remedy: Wait for search end.
Programm 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
Explanation: The function (2nd parameter) must not be activated during EXECUTE PROGRAM AREA.
Reaction: Alarm display.
Remedy: Wait for end of program area EXECUTE.
Programm 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
Explanation: 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.
Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

16963 [Channel %1:] ASUB start declined

Parameters: %1 = Channel number

Explanation: An external ASUB start fom the canceled 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.

Programm continuation: Clear alarm with the Delete key or NC START.

16964 [Channel %1:] Executing of init blocks not fully completed

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Switch control OFF - ON.

16966 [Channel %1:] Action '%2<ALNX>' prohibited during Jog Retract

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

Explanation: The function (2nd parameter) must not be activated during Jog Retract.

Reaction: Alarm display.

Remedy: End Jog Retract by Reset.

Programm continuation: Clear alarm with the Delete key or NC START.

16967 [Channel %1:] Action '%2<ALNX>' while preparing the protection areas not permissible

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

Explanation: The function (2nd parameter) must not be activated while the protection areas, which are activated using the PI _N_PROT_A, are being prepared.

Reaction: Alarm display.

Remedy: Wait until PI _N_PROT_A has been completed, or cancel PI with a RESET.

Programm continuation: Clear alarm with the Delete key or NC START.

17000 [Channel %1:] Block %2 maximum number of symbols exceeded

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

Explanation: 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)

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm continuation: Clear alarm with the RESET key. Restart part program

17018 [Channel %1:] Block %2 incorrect value for parameter %3

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

Explanation: An incorrect value has been assigned to the stated parameter.
Only the following values are permissible for the parameter $P_WORKAREA_CS_COORD_SYSTEM
=1 for workpiece coordinate system
=3 for settable zero system.

Reaction: Interpreter stop
Interface signals are set.
Alarm display.

Remedy: Assign another value.

Programm 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

Explanation: 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).
PROFIBUS I/O:
An illegal slot / I/O area index was used while reading/writing data.
Cause:
1.: Slot / I/O area index >= max. number of available slot / I/O areas.
2.: Slot / I/O area index references a slot / I/O area that has not been configured.
3.: Slot / I/O area index references a slot / I/O area that has not been released for system variables.
The following applies specifically: If the alarm occurs while writing one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR,
check whether MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC has been set correctly.
MD18077 $MN_MM_NUM_DIST_REL_PER_MAGLOC defines the number of different Index1 statements that may be made for an Index2 value.
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.

Programm 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

Explanation: General:
A read or write access has been programmed to an array variable with an invalid 2nd array index. The valid array indices must lie within the defined array size and the absolute limits (0 - 32,766).
PROFIBUS I/O:
An attempt was made to read/write data outside the slot / I/O area limits of the stated slot / I/O area.
The following applies specifically: If the alarm occurs while writing one of the parameters $TC_MDP1/$TC_MDP2/$TC_MLSR,
check whether MD18076 $MN_MM_NUM_LOCS_WITH_DISTANCE has been set correctly.
$MN_MM_NUM_LOCS_WITH_DISTANCE defines the number of different buffer storage locations that may be named in Index2.

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

Remedy: Correct the specification of array elements in the access instruction to match the defined size.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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])

- The value of MTL is incorrect. MTL must be programmed with a multitool location number of a multitool that is programmed in the command T=magazine location number in the same NC block.

- 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.

Programm 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

Explanation: 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].

Programm 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

Explanation: 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.
Programm 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
Explanation: 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

Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: An attempt was made to read or set a digital NC 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17140 [Channel %1:] Block %2 NC output %3 is assigned to a function via machine data

Parameters: %1 = Channel number
%2 = Block number, label
%3 = No. of output

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17150 [Channel %1:] Block %2 maximum of %3 NC outputs programmable in the block

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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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: --

Programm 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

Explanation: 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 part 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

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17182 [Channel %1:] Block %2 illegal sum correction number

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

Explanation: An attempt was made to access a non-defined total offset of the current tool edge.

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

Remedy: Access the total offset memory with $TC_SCP*, $TC_ECP*, check the total offset selection DLx or tool selection Ty or offset selection Dz.

Programm 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

Explanation: 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

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17184 [Channel %1:] Block %2 Tool %3, Duplo no. %5, D no.= %6 does not exist

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Tool identifier
%4 = Duplo number | D number

Explanation: A D number has been programmed that does not exist for the selected tool.

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

Remedy: If the selected tool does not contain the programmed D number,
* the data block for the cutting edge can be loaded into the NC (via HMI, with overstore) or
* the cutting edge can be created subsequently, or,
* if necessary, the D number and/or the tool identifier can be corrected in the displayed block,
and the NC program can be continued..

Programm 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

Explanation: 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.

Programm 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

Explanation: In the displayed block, a tool is accessed that is not defined and is therefore not available. The tool has been named by its T number, its name or its name and duplo number.
If the function T=magazine location is activated and programmed, the programmed T number indicates the magazine location number. The alarm can then indicate that the programmed magazine location number is illegal.
If the function T=magazine location is activated and programmed, and the multitool function is also activated and programmed in the program line MTL, the alarm can also indicate that there is no multitool at the location programmed with T=magazine location
or that there is no tool at the multitool location programmed with MTL.

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

Remedy: Check tool call in the NC part program:
- Is the correct tool number T.. programmed?
- T=magazine location programming: Is the correct magazine location number programmed?
- MTL programming: Is there a multitool at the programmed magazine location or is there a tool at the programmed multitool location?
- Are tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered previously either via the operator panel or 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

Programm 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

Explanation: A tool identifier which the NCK does not recognize was programmed.

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

Remedy: If the program pointer is at an NC block which contains the specified T identifier: If the program is incorrect, remedy the error with a correction block and continue the program. If the data block is missing, create one. You can do this by downloading a data block with all the defined D numbers onto the NCK (via HMI with overstore) and continue the program.
If the program pointer is at an NC block which does not contain the specified T identifier: The error occurred at an earlier point in the program where the T command appeared, but the alarm was not output until the change command was detected.
If the program is incorrect - T5 programmed instead of T55 - the current block can be corrected with a correction block; i.e. if only M06 is entered, you can correct the block with T55 M06. The incorrect T5 line remains in the program until it is terminated by a RESET or end of program.
In complex program structures with indirect programming, it may not be possible to correct the program. In this case, you can only intervene locally with an overstore block - with T55 in the example. If the data block is missing, create one. You can do this by downloading the data block of the tool with all the defined D numbers onto the NCK (via HMI with overstore), program the T command with overstore, and continue the program.

Programm 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

Explanation:
- 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.
- MTL=MT location T=magazine location number was programmed. There is no multitool at the programmed location

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?

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: The 'Hirth tooth system' function is activated for the indexing axis. This axis can therefore approach only indexing positions, a different travel movement of the axis is not possible.
It is not permissible to program PRESETON or PRESETONS.

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

17600 [Channel %1:] Block %2 preset on transformed axis %3 not possible

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

Explanation: The programmed preset key axis is involved in the current transformation. This means that setting the actual value memory (PRESET) is not possible for this axis (valid for PRESETON and PRESETONS).
Example:
Machine axis A should be set to the new actual value A 100 at the absolute position A 300.
:
N100 G90 G00 A=300
N101 PRESETON A=100

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

Remedy: Avoid preset actual value memory for axes which are participating in a transformation or deselect the transformation with the keyword TRAFOOF.

Programm continuation: Clear alarm with NC START or RESET key and continue the 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

Explanation: Actual value preset not possible for this axis because
- the axis is moving
- the axis is involved in a transformation
- the axis is currently being monitored for collision
- the spindle traverse movement has not yet been completed
- reference point approach (G74) is active
- the geometry axis is not in the 'neutral axis' state

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

Remedy: Avoid actual value preset or put the relevant geometry axis in the 'neutral axis' state in good time with RELEASE(...).

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17602 [Channel %1:] Block %2 motion synchronized action: %3 preset not possible on axis %4 in JOGREF mode

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name, spindle number

Explanation: The actual value cannot be preset for this axis if the mode group to which this axis is assigned is in JOG mode and if the machine function reference point approach is also selected.

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

Remedy: Avoid setting actual value in this configuration or deselect the machine function reference point approach.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17603 [Channel %1:] Block %2 motion synchronized action: %3 preset is only possible on a stationary axis %4 in JOG mode

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name, spindle number

Explanation: In JOG mode, an actual value can only be preset on stationary axes.

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

Remedy: Avoid setting an actual value while the axis is in motion; wait until the axis is stationary.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17604 [Channel %1:] Block %2 motion synchronized action: %3 preset not possible on active oscillating axis (axis %4)

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name, spindle number

Explanation: An actual value cannot be preset on an active oscillating axis controlled via synchronized actions.

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

Remedy: Avoid setting actual value during active oscillating movement.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17610 [Channel %1:] Block %2 axis %3 involved in the transformation, action cannot be carried out

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

Explanation: The axis is involved in the active transformation. It can therefore not execute the demanded action, traversing as positioning axis, enable for axis exchange.

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

Remedy: Deselect the transformation with TRAFOOF ahead of time or remove the action from the part program block

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17620 [Channel %1:] Block %2 approaching fixed point for transformed axis %3 not possible

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

Explanation: In the displayed block, an axis is programmed for the fixed point approach (G75) that is involved in the active transformation. Fixed point approach is not performed with this axis!

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

Remedy: Remove G75 instruction from the part program block or previously deselect transformation with TRAFOOF.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17630 [Channel %1:] Block %2 referencing for transformed axis %3 not possible

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

Explanation: In the displayed block, an axis is programmed for reference point approach (G74) that is involved in the active transformation. Reference point approach is not performed with this axis!

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

Remedy: Remove G74 instruction, or the machine axes involved in transformation, from the part program block or previously deselect the transformation with TRAFOOF.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

17650 [Channel %1:] Block %2 machine axis %3 not programmable

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

Explanation: The machine axis cannot be used in an active transformation. You may be able to program the function in a different coordinate system. For example, it may be possible to specify the retraction position in the basic coordinate system or the workpiece coordinate system. The axis identifier is used to select the coordinate system.

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

Remedy: Deactivate the transformation or use another coordinate system.

Programm 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

Explanation: 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[].

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: '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.

Programm 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

Explanation: '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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: '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.

Programm continuation: Clear alarm with the Delete key or NC START.

17822 [Channel %1:] Axis %2 JOG to position: Position changed

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

17823 [Channel %1:] Axis %2 JOG to position deactivated

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

17825 Indexing axis %1 $SA_JOG_POSITION unequal indexing position

Parameters: %1 = Axis number

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: The JOG in circles function 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.
20. JOG Retract is active. This mode is not supported.
Reaction: Interface signals are set.
Alarm display.
Remedy: Define axis as geometry axis.
Programm 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
Explanation: 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.
Programm 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
Explanation: At this point, the block context calls for a machine axis. This is the case with:
- G74 (reference point approach)
- G75 (fixed point approach)
- PRESETON/PRESETONS on GANTRY synchronous axis

If a geometry or special axis identifier is used, then it must also be permitted 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

18000 [Channel %1:] Block %2 NCK-specific protection area %3 wrong. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of NCK protection area
%4 = Error specification

Explanation: There is an error in the definition of the protection area. 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 area is not convex.
4: If both boundaries are active in the 3rd dimension of the protection area and both limits have the same value.
5: The number of the protection area does not exist (negative number, zero or greater than the maximum number of protection areas).
6: Protection area definition consists of more than 10 contour elements.
7: Tool-related protection area is defined as inside protection area.
8: Incorrect parameter used.
9: Protection area to be activated is not defined.
10: Incorrect modal G code used for protection area 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 area and check MD.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

18001 [Channel %1:] Block %2 channel-specific protection area %3 incorrect. Error code %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of the channel-specificprotection area
%4 = Error specification

Explanation: There is an error in the definition of the protection area. 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 area is not convex.

4: If both boundaries are active in the 3rd dimension of the protection area and both limits have the same value.

5: The number of the protection area does not exist (negative number, zero or greater than the maximum number of protection areas).

6: Protection area definition consists of more than 10 contour elements.

7: Tool-related protection area is defined as inside protection area.

8: Incorrect parameter used.

9: Protection area to be activated is not defined.

10: Incorrect modal G code used for protection area 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

18002 [Channel %1:] Block %2 NCK protection area %3 cannot be activated. Error code %4

Parameters: %1 = Channel number

%2 = Block number, label

%3 = Number of NCK protection zone

%4 = Error specification

Explanation: An error has occurred on activating the protection area. 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 area is not convex.

4: If both boundaries are active in the 3rd dimension of the protection area and both limits have the same value.

5: The number of the protection area does not exist (negative number, zero or greater than the maximum number of protection areas).

6: Protection area definition consists of more than 10 contour elements.

7: Tool-related protection area is defined as inside protection area.

8: Incorrect parameter used.

9: Protection area to be activated is not defined or number of contour element <2 or >MAXNUM_CONTOURNO_PROTECTAREA.

10: Error in internal structure of the protection areas.

11: Other, not further specified errors.

12: The number of simultaneously active protection areas exceeds the maximum number (channel-specific machine data).

13,14: Contour element cannot be created for protection areas.

15,16: No more memory space for the protection areas.

17: No more memory space for the contour elements.

18: The product of simultaneously activated or preactivated workpiece-related protection areas and tool-related protection areas is greater than 31.

Example: 16 workpiece-related and 1 tool-related protection area can be active.

The number of workpiece-related protection areas can then be increased, for example, to 19 (19 * 1 = 19).

However, the number of tool-related protection areas cannot be increased to 2 (16 * 2 = 32).

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 areas (MD).
2. Modify part program:
- Delete other protection areas.
- Preprocessing stop.
When the alarm occurs during control ramp-up, the system variables $SN_PA_... have to be corrected for the specified protection area. Afterwards perform a restart. If the erroneous data cannot be recognized, the protection area's immediate activation can be removed, and the system variables of the protection area can be written again by means of NPROTDEF.

Programm 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 area, the NC program must be canceled 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 area. 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 area %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

Explanation: An error has occurred on activating the protection area. 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 area is not convex.
4: If both boundaries are active in the 3rd dimension of the protection area and both limits have the same value.
5: The number of the protection area does not exist (negative number, zero or greater than the maximum number of protection areas).
6: Protection area definition consists of more than 10 contour elements.
7: Tool-related protection area is defined as inside protection area.
8: Incorrect parameter used.
9: Protection area 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 areas.
11: Other, not further specified errors.
12: The number of protection areas simultaneously active exceeds the maximum number (channel-specific machine data).
13,14: Contour element for protection areas cannot be created.
15,16: No more memory space for the protection areas.
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 areas (MD).
2. Modify part program:
- Delete other protection areas.
- Preprocessing stop.
When the alarm occurs on control ramp-up, system variables $SC_PA_... must be corrected for the specified protection area. Afterwards perform a restart. If the erroneous data cannot be recognized, the protection area's immediate activation can be removed, and the system variables of the protection area can be written again by means of CPROTDEF.

Programm 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 area, the NC program must be canceled 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 area. 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 area %3 does not correspond to the orientation of tool-related protection area %4

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Number of workpiece-related protection area

Explanation: The orientation of the workpiece-related protection area and the orientation of the tool-related protection area differ. If the protection area number is negative, then this is an NCK .

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

Remedy: - Modify the protection area definition or do not simultaneously activate protection areas that have different orientations.
- Check machine data and modify the protection area definition if necessary.

Programm 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 area %3

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

Explanation: The protection area definition must be terminated with EXECUTE before a preprocessing stop is performed.

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

Remedy: Modify part program.

Programm 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 area %3

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

Explanation: The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed.

Reaction: Correction block is reorganized.
Local alarm reaction.
Interface signals are set.
Alarm display.
Remedy: Modify part program.
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

18100 [Channel %1:] Block %2 invalid value assigned to FXS[]

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The following values are valid at the present time:
0: "Deselect traverse against fixed stop"
1: "Select traverse against fixed stop" valid.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

18101 [Channel %1:] Block %2 invalid value assigned to FXST[]

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Only the range 0.0 - 100.0 is valid at the present time.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with NC START or RESET key and continue the program.

18102 [Channel %1:] Block %2 invalid value assigned to FXSW[]

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Only positive values including zero are valid at the present time.
Reaction: Correction block is reorganized.
Interface signals are set.
Alarm display.
Remedy: --
Programm 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
Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.
Programm 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
Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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?

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

20005 [Channel %1:] Axis %2 reference point approach canceled

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Explanation: Referencing could not be completed for all stated axes (e.g., cancelation 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.

Value Meaning

-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.).

Programm continuation: Clear alarm with the RESET key. Restart part program

20006 [Channel %1:] Axis %2 reference point shutdown velocity not reached

Parameters: %1 = Channel number

%2 = Axis name, spindle number

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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)).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.
4. As part of a retraction motion (submode JOG-Retract) it cannot be traversed as a machine axis.

Reaction: Alarm display.

Remedy: Stop traversing through the channel or axis interface or stop the other geometry axis.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

20055 [Channel %1:] Master spindle not present in JOG mode

Parameters: %1 = Channel number

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 cancel with distance-to-go delete or RESET.

Programm 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

Explanation: The axis is currently not in "Geometry axis" state. Therefore, it cannot be traversed in JOG mode as geometry axis.
If the word Work (WCS = workpiece coordinate system) is displayed in the "Position" screen, 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

20070 [Channel %1:] Axis %2 software limit switch %3 programmed end position %4

Parameters: %1 = Channel number
%2 = Axis number
%3 = "1+" or "1-" for software limit switches 1, "2+" or "2-" for software limit switch 2,
%4 = Programmed end position

Explanation: The axis is to be traversed by the PLC as a concurrent positioning axis to the limit position. This would violate the corresponding software limit switch for the axis. No traversing takes place.
With an additional message to alarm 20140, the axis is to be traversed as a command axis.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Specify smaller target position. Modify MD for SW limit switch. Possibly activate another SW limit switch. Retract axis via JOG.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20071 [Channel %1:] Axis %2 working area limit %3 end position %4

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"
%4 = Programmed end position

Explanation: The displayed axis is to be traversed as a "concurrent positioning axis" to the programmed limit position and the corresponding working area limitation active for the axis is violated. No traversing takes place.
With an additional message to alarm 20140, the axis is traversed as a command axis.

Reaction: Alarm display.

Remedy: - Specify smaller target position.
- Deactivate working area limitation.
- Set working area limitation differently.
- Retract axis with JOG.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20072 [Channel %1:] Axis %2 is not an indexing axis

Parameters: %1 = Channel number
%2 = Axis number

Explanation: The displayed axis is operated as a concurrent positioning axis. Its target position is parameterized in the FC INDEX-AXIS as indexing position number, but the axis is not an indexing axis.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. The FC POS-AXIS for linear and rotary axes should be used or the axis should be declared as an indexing axis. Corresponding machine data for indexing axis declaration:
Modify MD30500 $MA_INDEX_AX_ASSIGN_POS_TAB
Modify MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1
Modify MD10910 $MN_INDEX_AX_POS_TAB_1
Modify MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2
Modify MD10930 $MN_INDEX_AX_POS_TAB_2

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20073 [Channel %1:] Axis %2 cannot be repositioned

Parameters: %1 = Channel number
%2 = Axis number

Explanation: The concurrent positioning axis cannot be positioned because it has already been restarted via the VDI interface and is still active. No repositioning motion takes place and the motion initiated by the VDI interface is not affected.

Reaction: Alarm display.

Remedy: None.

Programm continuation: Clear alarm with the Delete key or NC START.

20074 [Channel %1:] Axis %2 wrong index position

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

Explanation: For a concurrent positioning axis declared as indexing axis, the PLC has given an index number that is not available in the table.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Check the indexing axis number given by the PLC and correct this if necessary. If the indexing axis number is correct and the alarm results from an indexing position table that has been set too short, check the machine data for indexing axis declaration.
Modify MD30500 $MA_INDEX_AX_ASSIGN_POS_TAB
Modify MD10900 $MN_INDEX_AX_LENGTH_POS_TAB_1
Modify MD10910 $MN_INDEX_AX_POS_TAB_1
Modify MD10920 $MN_INDEX_AX_LENGTH_POS_TAB_2
Modify MD10930 $MN_INDEX_AX_POS_TAB_2

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20075 [Channel %1:] Axis %2 can currently not oscillate

Parameters: %1 = Channel number
%2 = Axis number

Explanation: The axis cannot perform an oscillating movement now because it is already being traversed, e.g. in JOG mode.

Reaction: Alarm display.

Remedy: End the other traversing motion.

Programm continuation: Clear alarm with the Delete key or NC START.

20076 [Channel %1:] Axis %2 oscillating - mode change not possible

Parameters: %1 = Channel number
%2 = Axis number

Explanation: The axis is performing an oscillating movement. Mode change is not possible because oscillation is not allowed in the selected mode.

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

Remedy: Please inform the authorized personnel/service department. Do not initiate mode change. Cause the PLC to check the axis and make sure in the PLC program that the axis ends oscillation if such mode changes take place.

Programm continuation: Clear alarm with the RESET key. Restart part program

20077 [Channel %1:] Axis %2 programmed position %4 is behind software limit switch %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"
%4 = Target position

Explanation: The axis is traversed as an oscillating axis and the target position (reversal position or end position) is located behind the corresponding software limit switch. The axis is not traversed.

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

Remedy: Specify smaller target position.
Modify MD for SW limit switch.
Possibly activate another SW limit switch.

Programm continuation: Clear alarm with the RESET key. Restart part program

20078 [Channel %1:] Axis %2 programmed position %4 is behind working area limit %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"
%4 = Target position

Explanation: The axis is traversed as an oscillating axis and the target position (reversal position or end position) is located behind the corresponding effective working area limitation. The axis is not traversed.

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

Remedy: Specify smaller target position.
Deactivate working area limitation.
Set working area limitation differentially.

Programm continuation: Clear alarm with the RESET key. Restart part program

20079 [Channel %1:] Axis %2 oscillation path %3 <= 0

Parameters: %1 = Channel number
%2 = Axis number
%3 = Length

Explanation: The axis is traversed as an oscillating axis and the distance to be traversed is smaller than or equal to zero. For example, both reversal points are situated on an identical position, one reversal point was shifted against the oscillating direction beyond the other reversal point. The axis is not traversed.

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

Remedy: Specify correct target position (reversal position, end position).

Programm continuation: Clear alarm with the RESET key. Restart part program

20080 [Channel %1:] Axis %2 no handwheel assigned for override

Parameters: %1 = Channel number
%2 = Axis number

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20081 [Channel %1:] Axis %2 braking position cannot be accepted as a new reversing position

Parameters: %1 = Channel number
%2 = Axis number

Explanation: On changing the reciprocation reversal from external sources, the braking position
cannot be accepted as a new reversing position, since changing the reversal point via handwheel or JOG key is active.

Reaction: Alarm display.

Remedy: Deselect VDI signal "Change reversal point" and reselect it either
- with "Reciprocation reversal from external sources" or
- by changing the reversal point by means of handwheel or
- by changing the reversal point via JOG key.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20082 [Channel %1:] Axis %2 coordinate system-specific working area limit %3 end position %4

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"
%4 = End position

Explanation: The displayed axis is operated as a "concurrent positioning axis", and the corresponding active coordinate system-specific working area limitation for the axis is violated. No traversing movement.
With an additional message to alarm 20140, the axis is traversed as a command axis.

Reaction: Alarm display.

Remedy: - Specify smaller target position.
- Deactivate working area limitation.
- Set working area limitation differently.
- Retract axis with JOG.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20083 [Channel %1:] Axis %2 programmed position %4 lies behind the coordinate system-specific working area limit %3

Parameters: %1 = Channel number
%2 = Axis number
%3 = "+" or "-"
%4 = End position

Explanation: The axis is traversed as a reciprocating axis, and the target position (reversal position or end position) is located behind the corresponding, valid, coordinate system-specific working area limitation. The axis is not traversed.

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

Remedy: Specify smaller target position.
Deactivate working area limitation.
Set working area limitation differentially.

Programm continuation: Clear alarm with the RESET key. Restart part program

20085 [Channel %1:] Contour handwheel: traverse direction or overtravel of beginning of block not allowed

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20090 Axis %1 travel to fixed stop not possible. Check programming and axis data.

Parameters: %1 = Axis name, spindle number

Explanation: 1. The "Traverse against fixed stop" function has been programmed with FXS[AX]=1 but the axis does not (yet) support this. Check MD37000 $MA_FIXED_STOP_MODE. This function is not available for gantry axes and simulated axes.
2. On selection, no movement was programmed for axis AX. AX is a machine axis identifier.
3. It is always necessary to program a traversing movement in the selection block for the axis/spindle for which the "Traverse against fixed stop" function is activated.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Please inform the authorized personnel/service department.
- Check the axis type.
- Check MD37000 $MA_FIXED_STOP_MODE.
- Is a machine axis movement missing in the approach block?

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

20091 Axis %1 has not reached fixed stop

Parameters: %1 = Axis name, spindle number

Explanation: On attempting to traverse against a fixed stop, the programmed end position has been reached or the traversing movement has been canceled. The alarm can be concealed by means of the MD37050 $MA_FIXED_STOP_ALARM_MASK.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Correct the part program and the settings:
- Has the traversing block been canceled?
- If the axis position does not correspond to the programmed end position, then correct the end position.
- If the programmed end position is in the part, the triggering criterion must be checked.
- Has the contour deviation leading to triggering been dimensioned too large? Has the torque limit been set too high?

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

20092 Axis %1 travel to fixed stop still active

Parameters: %1 = Axis name, spindle number

Explanation: An attempt has been made to move an axis while it is in fixed stop or while the deselection function has not yet been completed.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Please inform the authorized personnel/service department.
Check the following:
- Has the axis at the fixed stop also been moved by a traversing movement of geometry axes?
- Is a selection carried out even though the axis is stationary at the stop?
- Has the deselection process been interrupted by a RESET?
- Has the PLC switched the acknowledgment signals?

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

20093 Axis %1 standstill monitoring at fixed-stop end point has been triggered

Parameters: %1 = Axis name, spindle number

Explanation: The position of the axis has been beyond the zero speed window ever since selection has been completed.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Please inform the authorized personnel/service department.
- Check the mechanical components, e.g. has the stop broken away? Has the part to be clamped given way?
- Position window for zero speed control too small (MD37020 $MA_FIXED_STOP_WINDOW_DEF) (SD43520 $SA_FIXED_STOP_WINDOW). Default is 1 mm in each case.

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

20094 Axis %1 function has been canceled

Parameters: %1 = Axis name, spindle number

Explanation: The function has been canceled. The possible reasons for this are:
- The torque can no longer be provided because a pulse disable has occurred, .
- The PLC has removed the acknowledgments.

The system variable $VA_FXS_INFO contains additional information regarding why the function was canceled.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

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

Remedy: Check whether
- there is a pulse disable from the infeed/regenerative-feedback unit or from the PLC?
- the acknowledgment bits have been deleted by the PLC even though NCK has not requested deselection?

Read out the system variable $VA_FXS_INFO, and then interpret the additional information.

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

20120 Axis %1: too many compensation relations

Parameters: %1 = Axis name, spindle number

Explanation: Interpolatory compensation with tables. For each axis, the maximum number of compensation relationships defined may be no more than the number of axes in the system. In this alarm, the interpolatory compensation in the axis is switched off automatically.

Reaction: Interface signals are set.
Alarm display.

Remedy: Check table parameters $AN_CEC_OUTPUT_AXIS and correct and/or switch off one or more tables (SD41300 $SN_CEC_TABLE_ENABLE).

Programm continuation: Clear alarm with the RESET key. Restart part program

20121 Axis %1: Configuration error in compensation table %2

Parameters: %1 = Axis name, spindle number
%2 = Compensation table

Explanation: Interpolatory compensation with tables. The settings for the specified table are not allowed. $AN_CEC_MAX >= $AN_CEC_MIN and $AN_CEC_STEP != 0 apply to system variables. This table is switched off automatically.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Check and correct the characteristic data in the compensation table. If the error cannot be found, the alarm can be suppressed by switching off the table ($SN_CEC_TABLE_ENABLE) or switching off compensation in the axis ($MA_CEC_ENABLE).

Programm continuation: Clear alarm with the RESET key. Restart part program

20122 Compensation table %1: invalid axis assignment

Parameters: %1 = Compensation table

Explanation: Interpolatory compensation with tables. Assignment of the input or output axes in the given table is not allowed. $AN_CEC_INPUT_AXIS and $AN_CEC_OUTPUT_AXIS != 0 apply to system variables, and both must refer to valid axes. This table is automatically switched off.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Check and correct the axis assignment in the compensation table. If the error cannot be found, the alarm can be suppressed by switching off the table ($SN_CEC_TABLE_ENABLE) or switching off compensation in the axis ($MA_CEC_ENABLE).

Programm continuation: Clear alarm with the RESET key. Restart part program

20123 Axis %1: different output assignment of multiplied tables

Parameters: %1 = Axis name, spindle number

Explanation: Interpolatory compensation with tables. The two tables whose outputs are to be multiplied together have different output axes assigned to them. The compensation in this axis is automatically switched off.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Check and correct the characteristic data in the compensation table ($AN_CEC_OUTPUT_AXIS and $AN_CEC_MULT_BY_TABLE).
If the error cannot be found, the alarm can be suppressed by switching off the compensation in the axis ($MA_CEC_ENABLE) or the tables, ($SN_CEC_TABLE_ENABLE).

Programm continuation: Clear alarm with the RESET key. Restart part program

20124 Axis %1: sum of compensation values too large

Parameters: %1 = Axis name, spindle number

Explanation: The sum of the compensation values from all tables assigned to the axis had exceeded the limit value MD32720 $MA_CEC_MAX_SUM and had to be limited. Contour errors could have occurred as a result.

Reaction: Interface signals are set.
Alarm display.
Remedy: Check characteristic data of the compensation tables assigned to the axis.
Check characteristic curves in the tables ($AN_CEC).
Programm continuation: Clear alarm with the RESET key. Restart part program

20125 Axis %1: change of compensation value is too rapid

Parameters: %1 = Axis name, spindle number
Explanation: The compensation value has changed more rapidly than has been allowed for in MD32730 $MA_CEC_MAX_VELO. It had to be limited temporarily. The missing section is repeated later but contour errors might have occurred.
Reaction: Interface signals are set.
Alarm display.
Remedy: Check characteristic data of the compensation tables assigned to the axis.
Check characteristic curves in the tables ($AN_CEC). Possibly one of the input axes has moved more rapidly than provided for.
Programm 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
Explanation: The axis that should be traversed out of a synchronized action is not available
The following causes are possible:
- The axis is being or will be traversed by the NC program.
  This motion can also be made indirectly by means of continuous-path mode or an active frame.
- A superimposed motion is active for the axis.
- The axis is active as a following axis of a coupling.
- An interpolatory compensation, such as temperature compensation, is active for the axis.

Reaction: NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Remedy: Modify part program.
Programm 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
Explanation: 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.

Programm 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

Explanation: 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 HMI 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

20141 [Channel %1:] Block %2 motion synchronous action: %3 axis %4 has illegal axis type

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name

Explanation: The requested command is not permissible in the current axis status for the command axis or spindle. This alarm occurs with command axes (POS, MOV), spindle commands from motion synchronous actions (M3/M4/M5, SPOS), coupled motion (TRAILON, TRAILOF) and lead value coupling (LEADON, LEADOF).

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

Remedy: First stop the axis or deactivate the coupling, then select a new status.

Programm continuation: Clear alarm with the RESET key. Restart part program

20143 [Channel %1:] Block %2 motion synchronous action: %3 command axis %4 cannot be started as it is controlled by the PLC

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Axis name

Explanation: An attempt has been made to start a command axis by means of a block-related or modal synchronous action. This start is not possible as the axis is controlled by the PLC.

Reaction: Alarm display.

Remedy: End control of the axis by the PLC and therefore return it to the channel or start the command axis with a static synchronous action.

Programm continuation: Clear alarm with the Delete key or NC START.

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

Explanation: 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, for example, the required hardware components.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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-variable number.
PROFIBUS/PROFINET I/O:
An illegal slot / I/O area index was used while reading/writing data.
Cause:
1.: Slot / I/O area index >= max. number of available slots / I/O areas.
2.: Slot / I/O area index references a slot / I/O area that has not been configured.
3.: Slot / I/O area index references a slot / I/O area that has not been released for system variables.

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

Remedy: Use a valid index.

Programm continuation: Clear alarm with the RESET key. Restart part program

20170 [Channel %1:] Machine data $AC_FIFO invalid

Parameters: %1 = Channel number

Explanation: 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 variables 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 variables 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)

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

20302 [Channel %1:] Axis %2 cannot be traversed

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

Explanation: The displayed axis cannot be traversed as a machine axis because JOG Retract mode has been selected in JOG mode.

Reaction: Alarm display.

Remedy: Deselect JOG Retract with RESET

Programm continuation: Clear alarm with the Delete key or NC START.

20304 [Channel %1:] Axis %2 cannot be traversed as a geometry axis

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

Explanation: The displayed axis cannot be traversed as a geometry axis. The geometry axis is part of a retraction motion in JOG Retract mode. The requested motion of the geometry axis would lead to a violation of the permitted retraction direction.

Reaction: Alarm display.

Remedy: Deselect JOG Retract with RESET

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20306 [Channel %1:] Cartesian manual traverse not possible

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

Explanation: Cartesian manual traverse is not possible in JOG Retract mode.

Reaction: Alarm display.

Remedy: Deselect JOG Retract with RESET

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20308 [Channel %1:] Manual traverse in the SZS coodinate system is not possible

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

Explanation: Manual traverse in the SZS coodinate system is not possible in JOG Retract mode.

Reaction: Alarm display.

Remedy: Deselect JOG Retract with RESET

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

20310 [Channel %1:] Axis %2 traverse to the specified position is not possible

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

Explanation: The displayed axis cannot be traversed to the entered position in JOG Retract mode.
It is limited to the interrupt position when JOG Retract was selected.

Reaction: Alarm display.

Remedy: Traverse within the permitted positions

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

21550 [Channel %1:] Axis %2 Travel from hardware limit switch not possible. Reason: %3

Parameters: %1 = Channel number
%2 = Axis name
%3 = Cause

Explanation: 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

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

21600 Monitoring for ESR active

Explanation: -

Reaction: NC not ready.
Alarm display.
All alarm responses are delayed by one IPO cycle.

Remedy: The display can be suppressed with MD 11410 $MN_SUPPRESS_ALARM_MASK Bit 16 = 1

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

21610 [Channel %1:] Axis %2 encoder %3 frequency exceeded

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

Explanation: 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).

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

21611 [Channel %1:] NC-controlled Extended Stop/Retract triggered

Parameters: %1 = Channel number

Explanation: "NC-controlled Extended Stop/Retract" triggered.

Reaction: The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
All channel-specific alarm reactions are delayed with this alarm, alarm display.

Remedy: Reset

Programm continuation: Clear alarm with the RESET key. Restart part program

21612 [Channel %1:] Axis %2: enable reset, cause %3

Parameters: %1 = Channel number
%2 = Axis name, spindle number
%3 = Cause of the alarm

Explanation: 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 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 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).

Programm continuation: Clear alarm with the Delete key or NC START.

21613 Axis %1 measuring system changing

Parameters: %1 = Axis name, spindle number

Explanation: The measuring system for this axis is changing.

Reaction: Alarm display.

Remedy: -

Programm 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 +/-)

Explanation: 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.

Programm 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

Explanation: 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: -

Programm continuation: Clear alarm with the RESET key. Restart part program

21616 [Channel %1:] Block %2 overlaid motion active at transformation switchover

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

Explanation: The overlaid motion in the BCS changes its significance because of the transformation change and can therefore lead to undesired axis movements.

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

Remedy: Take out the overlaid movement.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

21617 [Channel %1:] Block %2 transformation does not allow to traverse the pole

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

Explanation: The preset curve passes through the pole or a forbidden area of the transformation.

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

Remedy: Modify the part program (if the alarm has occurred in AUTO mode).
To escape from the alarm position, transformation must be deselected (it is not enough to try a RESET if the transformation remains active when RESET is applied).

Programm continuation: Clear alarm with the RESET key. Restart part program

21618 [Channel %1:] As from block %2 transformation active: overlaid motion too great

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

Explanation: The share of overlaid motion on the transformation-related axes is so high that the path movement planned by the preparation no longer sufficiently corresponds to the actual ratio for the interpolation. Strategy of singularities, monitoring of working range limitation and dynamic Look Ahead are possibly no longer correct.

Reaction: Alarm display.

Remedy: With overlaid motion it is necessary to keep a sufficiently large path safety distance with regard to poles and working range limitations.

Programm continuation: Clear alarm with the Delete key or NC START.

21619 [Channel %1:] Block %2 transformation active: motion not possible

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

Explanation: The machine kinematics does not allow the specified motion.
Transformation-dependent error causes can be in:
TRANSMIT: A (circular) area exists around the pole, where positioning is not possible. The area is caused by the fact that the tool reference point cannot be traversed as far as into the pole. The area is defined by:
- the machine data (MD249.. $MC_TRANSMIT_BASE_TOOL_...)
- the active tool length compensation (see $TC_DP..).
Whether the tool length compensation is included in the calculation depends on the working plane selected (see G17,..). The machine stops at the edge of the area where positioning is not possible.

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

Remedy: Modify part program. Change the incorrectly specified tool length compensation.
Note: RESET alone is not enough if transformation also remains active during RESET.

Programm 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

Explanation: An axial emergency braking ramp was activated for the specified axis/spindle
The following causes are possible for the activation of the emergency braking ramp:
Alarm 26052: Path velocity for auxiliary function output too high
Alarm 1012 : System error with ID 550006 550006
Alarm 1016 : System error with ID 550003, 550005 and 550010
With master/slave coupling, MD30132 $MA_IS_VIRTUAL_AX (axis is virtual axis) has been set.
Braking request with priority 13 may have been requested by the OEM application.

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.

Programm continuation: Clear alarm with the RESET key. Restart part program

21621 [Channel %1:] Block %2 The constant transformation axis %3 has moved.

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

Explanation: An axis whose position must remain constant for the transformation in the current block has moved.

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

Remedy: Change part program if alarm has occurred in AUTO mode.
The alarm is cancelled with RESET. If the transformation is retained beyond RESET, the constant axis goes into the transformation with its new position. If that is not wanted, the transformation must be deselected, and the axis position changed in this state. The transformation can then be reactivated.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

21665 [Channel %1:] $AA_TOFF cleared

Parameters: %1 = Channel number

Explanation: If the tool direction 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.

Programm 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

Explanation: 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()

Programm continuation: Clear alarm with NC START or RESET key and continue the program.

21675 [Channel %1:] Block %2 impermissible motion on change of tool direction and $AA_TOFF active

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

Explanation: If an offset is active in tool direction via $AA_TOFF[i] there must not be a geometry axis movement in a block in which the tool orientation is changed abruptly.
Abrupt changes of the tool orientation, whereby a movement of the geometry axes can be programmed simultaneously, can, for example, occur during a plane change, tool change, or activation and deactivation of an orientable tool carrier.

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()

Programm 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

Explanation: 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?

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

21702 [Channel %1:] Block %3 axis %2 measurement canceled

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

Explanation: 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 axes are not identical.
When making measurements without handshake, two probes must be available in drive parameters p0680 index 0 and 1.

Reaction: Alarm display.

Remedy: Check the traversing movement in the measurement 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?
- Is the probe correctly wired and configured on the hardware side (e.g. drive parameters p0488 and p0489)?

Either program all GEO axes explicitly or program the traversing movement with the POS[axis] command.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

21800 [Channel %1:] Workpiece setpoint %2 reached

Parameters: %1 = Channel number
%2 = Workpiece setpoint

Explanation: 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.

Programm 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

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

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.

Programm 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

Explanation: 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 M45 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

22006 [Channel %1] Block %2 Motion-synchronous action: %3 Spindle %4 gear stage change not possible

Parameters: %1 = Channel number
%2 = Block number, line number
%3 = Synact ID
%4 = Spindle number

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

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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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 on the alarm generating function.
If this has been intentionally programmed, the alarm can be suppressed in the cyclic check with position control already enabled with following and leading spindle(s) by means of MD11410 $MN_SUPPRESS_ALARM_MASK bit21 = 1 or by CP programming with CPMALARM[FAx] bit10 = 1 (in which FAx = following spindle).

Programm 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

Explanation: - 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

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

22200 [Channel %1:] Block %3 Spindle %2 axis stopped during tapping

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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

22250 [Channel %1:] Block %3 Spindle %2 axis stopped during thread cutting

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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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).

Programm 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

Explanation: 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).

Programm 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

Explanation: 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).

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

22292 [Channel %1:] Changeover of the operating mode of the axis/spindle %2 using PI service not possible, Cause %3

Parameters: %1 = Channel number
%2 = Axis/spindle
%3 = Cause

Explanation: It is not possible to change over the operating mode of the axis/spindle.
Cause:
1: The required axis/spindle is not known in the channel.
2: The required axis/spindle is not available in the channel.
3: The required axis/spindle is defined as virtual axis.
4: The required axis is not defined as spindle. Therefore, it is not possible to change over the operating mode of the axis.
5: The required axis/spindle is a permanently assigned PLC axis/spindle.
6: The required axis/spindle is an active slave axis/spindle.
7: Spindle operation is not possible for transformed spindle/axis.
8: The required axis/spindle is not available as command axis.

Reaction: Alarm display.

Remedy: 1: Please select the PI service again.
2: Please select the PI service again.
3: Reset MD 30132 IS_VIRTUAL_AX.
4: When required, configure axis as spindle.
5: Axis/spindle enabled by the PLC.
6: Deactivate coupling of the slave axis/spindle.
7: Deactivate transformation.
8: Wait until the axis is again available.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

22324 [Channel %1:] Block %2 Axis %3 PRESETON or PRESETONS not permitted (cause: %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
%4 = Index describing the problem that occured in more detail

Explanation: A problem has occurred when programming PRESETON or PRESETONS. This problem is described in more detail using the "Index" parameter:
Index == 1: In MD30460 $MA_BASE_FUNCTION_MASK the PRESETON or PRESETONS function is deactivated.
Index == 2: For PRESETONS, the axis is monitored for collision. Actual value setting is not permitted.
Index == 3: PRESETON/PRESETONS is not permissible for synchronous GANTRY axes.
Index == 4: In speed control mode, PRESETON/PRESETONS is not permissible before terminating the traversing movement.

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

Remedy: Apply the programming permitted.
Please notify the authorized personnel/service.

Programm continuation: Clear alarm with the RESET key. Restart part program

22400 [Channel %1:] Option 'contour handwheel' not set

Parameters: %1 = Channel number

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

25000 Axis %1 hardware fault of active encoder

Parameters: %1 = Axis name, spindle number

Explanation: The signals of the currently active actual position 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).
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing PowerOn alarm by the Reset alarm 25010.

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

Remedy: Please inform the authorized personnel/service department. Check measuring circuit connectors for correct contacting. Check encoder signals and replace the encoder if faults are found.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

Programm continuation: Switch control OFF - ON.

25001 Axis %1 hardware fault of passive encoder

Parameters: %1 = Axis name, spindle number

Explanation: The signals from the currently inactive actual position value encoder are missing, or they are not of the same phase, or they exhibit grounding/short-circuit.
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 replaces the existing PowerOn alarm by the Reset alarm 25011.

Reaction: Alarm display.

Remedy: Please inform the authorized personnel/service department. Check that correct contacts are established for the measuring circuit connectors. 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).
The fault cause remains until the next power on
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1, 2).

Programm continuation: Switch control OFF - ON.

25010 Axis %1 pollution of measuring system

Parameters: %1 = Axis name, spindle number

Explanation: The encoder used for position control sends a contamination signal (only in measuring systems with contamination signal).
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).
For PROFIdrive only:
MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Reset alarm instead of the Power-on alarm 25000.

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

Remedy: Please inform the authorized personnel/service department. Check the measuring system in accordance with the instructions given by the measuring device manufacturer.
Monitoring can be switched off by setting MD36310 $MA_ENC_ZERO_MONITORING[n] to 100 (n = encoder number: 1,2).

Programm continuation: Restart part program. 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

Explanation: The encoder not used for position control sends a contamination signal (only in measuring systems with contamination signal).

For PROFIdrive only:

MD36310 $MA_ENC_ZERO_MONITORING >100 returns the existing Reset alarm instead of the Power-on alarm 25001.

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).

Programm continuation: Clear alarm with the RESET key. Restart part program

25020 Axis %1 zero mark monitoring of active encoder

Parameters: %1 = Axis name, spindle number

Explanation: For PROFIdrive:

The position encoder pulses between 2 zero mark pulses are counted, and the plausibility is assessed (The functionality and possibly the parameterization of the plausibility check is done on the drive side. Please refer to the relevant drive documentation for details.), and reported in a PROFIdrive message frame (encoder interface) to the control, which then issues the present alarm.

The alarm can be reprogrammed in MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.

The NC switches to follow-up mode.

Channel not ready.

NC Start disable in this channel.

Axes of this channel must be re-referenced.

Interface signals are set.

Alarm display.

NC Stop on alarm.

Channel not ready.

Remedy: Please inform the authorized personnel/service department.

The differences can result from transmission errors, disturbances, encoder hardware faults or from the evaluation electronics in the encoder used for position control. The actual value branch must therefore be checked:

1. Transmission path: Check the actual-value connectors for correct contacting, encoder cable for continuity, and also check for short-circuits and grounding (loose contact?).

2. Encoder pulses: Is the encoder power supply within the tolerance limits?

3. Evaluation electronics: Replace or reconfigure the drive or encoder module used.

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).

Programm continuation: Restart part program. 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

Explanation: 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).

Programm 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

Explanation: 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. Only on the SIMODRIVE 611D if 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).

Programm continuation: Clear alarm with the Delete key or NC START.

25030 Axis %1 actual velocity alarm limit

Parameters: %1 = Axis name, spindle number

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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: -

Programm continuation: Clear alarm with the Delete key or NC START.

25040 Axis %1 standstill monitoring

Parameters: %1 = Axis name, spindle number

Explanation: 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).

Programm continuation: Restart part program. 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

Explanation: 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 canceled 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).

Programm continuation: Restart part program. 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

Explanation: 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%).

For SINAMICS: Drive parameter p1082 also has a limiting effect.

If the values are exceeded for a short time, then this is tolerated provided they do not last longer than allowed for in the axis-specific MD36220 $MA_CTRLOUT_LIMIT_TIME. The setpoint is limited during this time to the maximum value that has been set in (MD36210 $MA_CTRLOUT_LIMIT).

The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.

The NC switches to follow-up mode.

Channel not ready.

NC Start disable in this channel.

Interface signals are set.

Alarm display.

NC Stop on alarm.

Channel not ready.

Remedy: Please inform the authorized personnel/service department. This alarm should not occur if the drive controller has been set correctly and the machining conditions are those that normally prevail.

- Check actual values: Local sluggishness of the carriage, speed dip by torque surge due to contact with workpiece/tool, travel against fixed obstacle, etc.

- Check direction of position control: Does the axis continue to rotate without control?

Programm continuation: Restart part program. 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

Explanation: 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).

Programm continuation: Clear alarm with the Delete key or NC START.

25080 Axis %1 positioning monitoring

Parameters: %1 = Axis name, spindle number

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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 actual position 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 actual position 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 canceled with "RESET", then program execution can be reinitiated with NC START, if necessary at the interruption point after "Block search with/without calculation".

Programm continuation: Clear alarm with the RESET key. Restart part program

25105 Axis %1 measuring systems differ considerably

Parameters: %1 = Axis name, spindle number

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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

Programm continuation: Clear alarm with the Delete key or NC START.

25200 Axis %1 requested set of parameters invalid

Parameters: %1 = Axis name, spindle number

Explanation: 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]
- MD32452 $MA_BACKLASH_FACTOR [n]
- MD32610 $MA_VELO_FFW_WEIGHT [n]
- MD32800 $MA_EQUIV_CURRCTRL_TIME [n]
- MD32810 $MA_EQUIV_SPEEDCTRL_TIME [n]
- MD32910 $MA_DYN_MATCH_TIME [n]
- MD36012 $MA_STOP_LIMIT_FACTOR [n]
- MD36200 $MA_AX_VELO_LIMIT [n]

Programm continuation: Clear alarm with the RESET key. Restart part program

25201 Axis %1 drive fault

Parameters: %1 = Axis name, spindle number

Explanation: For PROFIdrive:
The drive signals a serious fault which prevents the drive from being ready. The exact cause of the fault can be found by evaluating the output drive alarms (It may be necessary to activate these diagnostic alarms by parameterizing the MDs MD10070 $MN_DRIVE_FUNCTION_MASK, MD13140 $MN_PROFIBUS_ALARM_ACCESS etc):
Alarms 380500 and 380501 (or the corresponding alarm numbers implemented on the HMI side).
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Channel not ready.

Remedy: Evaluation of the drive alarms.

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

25202 Axis %1 waiting for drive

Parameters: %1 = Axis name, spindle number

Explanation: Drive group error (self-clearing).

Reaction: Interface signals are set.
Alarm display.

Remedy: For PROFIdrive only:
Wait for the drive. This alarm reveals similar problems to alarm 25201 (see that alarm). It is continuously active during power-up if the drive does not communicate (e.g. if the PROFIBUS connector has fallen out). Otherwise, the alarm is active only briefly and is replaced by alarm 25201 after an internal timeout in the event of a permanent problem.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

25220 Axis %1 gear ratio changed with ESR enabled

Parameters: %1 = Axis name, spindle number

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

26000 Axis %1 clamping monitoring

Parameters: %1 = Axis name, spindle number

Explanation: 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).

Programm continuation: Restart part program. 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

Explanation: 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.

Programm continuation: Restart part program. 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 pulses

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

Explanation: 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 (PROFIdrive: p979) or one of the two values is zero.
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.
For PROFIdrive drives:
Compare drive parameter p979 (and possibly other internal drive, manufacture-specific parameters stated in the relevant drive documentation) or compare images in NC-MDs $MA_ENC_RESOL, $MA_ENC_PULSE_MULT, $MA_ENC_ABS_TURNS_MODULO etc.
Conditions leading to triggering the alarm are:
* Number of encoder pulses in the drive!= $MA_ENC_RESOL
* PROFIdrive interface standardization/high resolution in p979 not permitted (permitted pushing factor 0...30 bits)
* With absolute encoders: Interface formats in p979 for absolute and incremental information does not fit together (i.e. the absolute position in XIST2 is supplied in too coarse resolution for a complete position reconstruction)
* In the case of rotary absolute encoders behind the gears (and active traversing range extension in accordance with $MA_ENC_ABS_BUFFERING): Absolute position format (in Gx_XIST2) is not complete/sufficient for position reconstruction via PowerOff in accordance with the following condition: MA_ENC_RESOL*MA_ENC_PULSE_MULT* $MA_ENC_ABS_TURNS_MODULO must not be smaller than 2**32. A remedy against alarm triggering in the latter case may be to increase $MA_ENC_PULSE_MULT (or the associated drive-side parameterization e.g. p418/419 with SINAMICS) or (when the necessary conditions prevail) by deactivating the traversing range extension - see $MA_ENC_ABS_BUFFERING

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.

Programm continuation: Switch control OFF - ON.

26003 Axis %1 parameterization error: lead screw pitch

Parameters: %1 = Axis name, spindle number

Explanation: 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.).

Programm 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

Explanation: The scale division of the linear scale set in the axis-specific MD31010 $MA_ENC_GRID_POINT_DIST is zero or differs from the corresponding drive parameters. For a better understanding of the interrelations see the explanations for alarm 26002, which refer to rotatory encoders.
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Channel not ready.

Remedy: Please inform the authorized personnel/service department.
Enter the encoder grid point distance according to the data given by the machine (or measuring device) manufacturer in the MD31010 $MA_ENC_GRID_POINT_DIST.

Programm continuation: Switch control OFF - ON.

26005 Axis %1 parameterization error: output rating

Parameters: %1 = Axis name, spindle number

Explanation: 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 effective output evaluation of the speed setpoint interface is zero:
a. MD32260 $MA_RATED_VELO is zero although a standardizing reference value needs to be defined on account of the setting MD32250 $MA_RATED_OUTVAL>0
b. The corresponding drive-side standardizing parameter is zero, invalid or unreadable/unavailable although an automatic interface scaling adjustment has been selected on account of MD32250 $MA_RATED_OUTVAL=0.
The drive parameter defining the standard is not determined by PROFIdrive, but is specific to the manufacturer (see the relevant drive documentation: For SINAMICS: p2000).
The alarm can be reprogrammed in the MD11412 $MN_ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.
Channel not ready.

Remedy: Please inform the authorized personnel/service department. The nominal output voltage in [%] of the maximum setpoint value (10 V) is entered in the MD32250 $MA_RATED_OUTVAL, at which the rated motor speed in [degrees/s] is to be reached (MD32260 $MA_RATED_VELO).

Programm continuation: Restart part program. 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

Explanation: 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 (PROFIdrive)
= 4 Absolute encoder (all drive-side absolute encoders supported by PROFIdrive)
MD30130 $MA_CTRLOUT_TYPE
= 0 Simulation
= 1 Standard (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.

Programm continuation: Switch control OFF - ON.

26014 Axis %1 machine data %2 invalid value

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

Explanation: 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.

Programm 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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

26016 Axis %1 machine data %2 invalid value

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

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: 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.

Programm continuation: Restart part program. 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

Explanation: The same setpoint assignment has been allocated more than once.
MD30110 $MA_CTRLOUT_MODULE_NR contains the same value for different axes.
PROFIdrive: The stated MDs contain the same values for different axes, or different entries in $MN_DRIVE_LOGIC_ADDRESS contain the same values.

Reaction: Mode group not ready.
The NC switches to follow-up mode.
Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.
NC Stop on alarm.

Remedy: Please inform the authorized personnel/service department. Avoid dual assignment of the setpoint by correcting MD30110 $MA_CTRLOUT_MODULE_NR. Also check the selected bus type MD30100 $MA_CTRLOUT_SEGMENT_NR.

Programm 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

Explanation: 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].

Programm continuation: Clear alarm with the RESET key. Restart part program

26020 Axis %1 encoder %2 hardware fault %3 during encoder initialization

Parameters: %1 = Axis name, spindle number
%2 = Encoder number
%3 = Error fine coding

Explanation: Error during initialization or access of encoder.

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

Remedy: Please inform the authorized personnel/service department. Rectify hardware error, replace encoder if necessary.

Programm continuation: Switch control OFF - ON.

26022 Axis %1 encoder %2 measurement with simulated encoder not possible

Parameters: %1 = NC axis number
%2 = Encoder number

Explanation: 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).

Programm 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

Explanation: The machine data contains an invalid value and therefore has been changed by the software.

Reaction: Alarm display.

Remedy: Check MD.

Programm 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

Explanation: 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.

Programm continuation: Clear alarm with the RESET key. Restart part program

26026 Axis %1 SINAMICS drive parameter P2038 value is not allowed.

Parameters: %1 = Axis name, spindle number

Explanation: For SINAMICS drives only:
The interface mode, which is set via drive parameter P2038, has not been set to SIMODRIVE 611 universal.
The alarm can be disabled by MD13070 $MN_DRIVE_FUNCTION_MASK - bit15.
However, the following must be noted:
- The device-specific assignment of the bits in the control and status words may be different.
- The drive data sets can be created at will, and need not be subdivided into groups of 8. (For details see also SINAMICS Commissioning Manual). So the parameters of motors 2-4 may be incorrectly assigned.

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

Remedy: - Set P2038 = 1 or
- Set P0922 = 100...199 or
- Set bit 15 of MD13070 $MN_DRIVE_FUNCTION_MASK (note the boundary conditions, see above)
and execute a Power ON in each case.

Programm continuation: Switch control OFF - ON.

26027 Axis %1 Stiffness mode Splines is not available (%2)

Parameters: %1 = Axis name, spindle number
%2 = Fine ID

Explanation: The Stiffness mode Splines is not available.
Fine coding:
Bit 0 - SINAMICS function block is not present (see P0108)
Bit 1 - $MN_DRIVE_TELEGRAMM_TYPE, set telegram that supports splines functionality (e.g. telegram 136)
Bit 2 - Set $MA_VELO_FFW_WEIGHT = 100%
Bit 3 - Set $MA_FIPO_TYPE = 2
Bit 4 - Set $MA_FFW_MODE = 3 or 4
Bit 5 - Set $MN_POSCTRL_SYSCLOCK_TIME_RATIO = 1

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

Remedy: Set $MA_SPLINES_CONTROL_CONFIG=0, or note and implement fine error code.

Programm continuation: Switch control OFF - ON.

26028 Machine data %1[%2] value not permissible

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

26030 Axis %1 encoder %2 absolute position lost

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

Explanation: 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 was changed, see MD34230 $MA_ENC_SERIAL_NUMBER, and drive-specific parameters).
- of an internal number format limitation in MD34090 $MA_REFP_MOVE_DIST_CORR. Remedy: reduce MD10210 $MN_INT_INCR_PER_DEG or MD10200 $MN_INT_INCR_PER_MM.

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).

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

26040 Axis %1 encoder parameterization MD %2[%3] has been adjusted.

Parameters: %1 = Axis number
%2 = MD identifier
%3 = Machine data index

Explanation: The parameterization of the encoder in P979 read out from the drive and displayed in the MD does not match the NCK parameterization. The corresponding NCK-MD has been adapted.

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: Power on required. This alarm only occurs when the NC-MD is set unequal to the drive parameter.

Programm continuation: Switch control OFF - ON.

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

Explanation: 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).

Programm 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

Explanation: The path interpolation did not stop, as required, at the end of the block, but only decelerates to a standstill in the next block. This error situation occurs if the stop at the end of the block was not planned by the path interpolation or was not detected early enough. One 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 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.

Programm continuation: Clear alarm with the Delete key or NC START.

26052 [Channel %1:] In block %2: path velocity too high for auxiliary function output (%3)

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

Explanation: This alarm usually occurs in a block with auxiliary function output during a movement. In this case, the wait for acknowledgment 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). Alarm 21620 is often triggered as a follow-up alarm. If not, the path continues after the block change.

Reaction: Alarm display.

Remedy: - The time calculated for the auxiliary function output during the movement comes from the PLC in most systems. Otherwise, MD10110 $MN_PLC_CYCLE_TIME_AVERAGE is used for this purpose.
- The alarm can basically be avoided by programming in the block G09 indicated in the message. This stops the path interpolation briefly at the end of the block.

Programm continuation: Clear alarm with the Delete key or NC START.

26053 [Channel %1:] Block %2 interpolation problem in Look Ahead (module %3, identifier %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Module identifier
%4 = Error code

Explanation: Synchronism between interpolation and preparation is faulty.

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

Remedy: Please contact Siemens.

Programm continuation: Clear alarm with the RESET key. Restart part program

26054 [Channel %1:] Block %2 interpolation warning in Look Ahead (module %3, problem %4)

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Module identifier
%4 = Error code

Explanation: The computer performance is inadequate to create a smooth path velocity profile. This can lead to drops in velocity.

Reaction: Local alarm reaction.
Alarm display.
Warning display.

Remedy: Change parameterization. Increase interpolation cycle.

Programm continuation: Clear alarm with the Delete key or NC START.

26070 [Channel %1:] Axis %2 cannot be controlled by the PLC, max. number exceeded

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

Explanation: An attempt was made to define more axes than allowed as PLC-controlled axes.

Reaction: Interface signals are set.
Alarm display.

Remedy: Check the option 'Number of PLC-controlled axes' and correct if necessary or reduce the number of requests for PLC-controlled axes.

Programm continuation: Clear alarm with the Delete key or NC START.

26072 [Channel %1:] Axis %2 cannot be controlled by the PLC

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

Explanation: Axis cannot be made a PLC-controlled axis. For the time being, the axis cannot be controlled at any state from the PLC.

Reaction: Interface signals are set.
Alarm display.

Remedy: Use Release or Waitp to make the axis a neutral one.

Programm continuation: Clear alarm with the Delete key or NC START.

26074 [Channel %1:] Switching off PLC control of axis %2 not allowed in the current state

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

Explanation: The PLC can return the control rights for an axis to program processing only, if the axis is in READY state.

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

Remedy: Reset VDI interface signal "PLC controls axis", then activate "Axial reset" and repeat process.

Programm continuation: Clear alarm with the Delete key or NC START.

26075 [Channel %1:] Block %2 Axis %3 not available for the NC program, as exclusively controlled by the PLC

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

Explanation: The axis is exclusively controlled by the PLC. Therefore, the axis is not available for the NC program.

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

Remedy: Have the axis not exclusively controlled by the PLC, but only temporarily. Change MD30460 $MA_BASE_FUNCTION_MASK, bit 4.

Programm continuation: Clear alarm with the RESET key. Restart part program

26076 [Channel %1:] Block %2 Axis %3 not available for NC program, firmly assigned PLC axis

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

Explanation: The axis is a firmly assigned PLC axis. The axis is therefore not available for the NC program.

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

Remedy: Do not define axis as a firmly assigned PLC axis. Change of MD30460 $MA_BASE_FUNCTION_MASK bit5.

Programm continuation: Clear alarm with the RESET key. Restart part program

26077 [Channel %1:] Axis %2 not available for the NC program, as exclusively controlled by the PLC

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

Explanation: The axis is exclusively controlled by the PLC. Therefore, the axis is not available for the NC program.

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

Remedy: Have the axis not exclusively controlled by the PLC, but only temporarily. Change MD30460 $MA_BASE_FUNCTION_MASK, bit 4.

Programm continuation: Clear alarm with the RESET key. Restart part program

26078 [Channel %1:] Axis %2 not available for NC program, firmly assigned PLC axis

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

Explanation: The axis is a firmly assigned PLC axis. The axis is therefore not available for the NC program.

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

Remedy: Do not define axis as a firmly assigned PLC axis. Change of MD30460 $MA_BASE_FUNCTION_MASK bit5.

Programm continuation: Clear alarm with the RESET key. Restart part program

26080 [Channel %1:] Retraction position of axis %2 not programmed or invalid

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

Explanation: No retraction position has been programmed for the axis trigger time or the position became invalid.

Reaction: Alarm display.

Remedy: Preset value by means of POLFA(Axis,Type,Pos), with type = 1 (absolut) or type = 2 (incremental); type = 0 specifies the position as invalid.

Programm continuation: Clear alarm with the Delete key or NC START.

26081 [Channel %1:] Axis trigger of axis %2 was activated, but axis is not PLC-controlled

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

Explanation: The axis trigger for single axis was initiated. However, the axis is not PLC-controlled at the trigger time (therefore no single axis) or the position became invalid.

Reaction: Alarm display.

Remedy: Preset axis PLC-controlled (declare single axis).

Programm continuation: Clear alarm with the Delete key or NC START.

26082 [Channel %1:] ESR for PLC-controlled axis %2 has been triggered

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

Explanation: An axial ESR has been triggered for an individual axis (PLC-controlled axis):
The display can be suppressed by MD11410 $MN_SUPPRESS_ALARM_MASK bit28 = 1.

Reaction: Alarm display.

Remedy: The individual axis is in axial stop after the ESR movement.
If an axial reset is performed for the individual axis, the alarm will be deleted
and the individual axis can be traversed again.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.
The individual axis is in axial stop after the ESR movement.
If an axial reset is performed for the individual axis, the alarm will be deleted
and the individual axis can be traversed again.

26100 Axis %1 drive %2 sign of life missing

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

Explanation: 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.

Programm continuation: Switch control OFF - ON.

26101 Axis %1 drive %2 communication failure

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

Explanation: For PROFIdrive only:
The drive is not communicating.

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

Remedy: - Check the bus configuration.
- Check the interface (connector removed, option module inactive, etc.).

Programm continuation: Clear alarm with the RESET key. Restart part program

26102 Axis %1 drive %2 sign of life missing

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

Explanation: For PROFIdrive only:
The sign-of-life cell is no longer being updated by the drive.

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

Remedy: - Check the cycle settings (recommendation: e.g. MD10062 $MN_POSCTRL_CYCLE_DELAY = 0.0)
- Increase the cycle time if necessary.
- Power-up the drive again.
- Check drive software.

Programm continuation: Clear alarm with the RESET key. Restart part program

26105 Drive of axis %1 not found

Parameters: %1 = Axis name, spindle number

Explanation: For PROFIdrive only:

The drive configured for the specified axis could not be found. For example, a PROFIBUS slave was configured on the NC but is not contained in SDB-Type-2000.

Reaction: Mode group not ready.

The NC switches to follow-up mode.

Channel not ready.

NC Start disable in this channel.

Interface signals are set.

Alarm display.

NC Stop on alarm.

Remedy: Possible causes:

- MD30130 $MA_CTRLOUT_TYPE not equal to 0 as a result of an oversight; the drive should actually be simulated (= 0).
- MD30110 $MA_CTRLOUT_MODULE_NR entered incorrectly, i.e. the logical drive numbers were exchanged and an invalid value is stored for this drive in MD13050 $MN_DRIVE_LOGIC_ADDRESS (see 3.) or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- MD13050 $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the PROFIBUS (i.e. the values are not in SDB-Type-2000) or different addresses were selected for the input and output slots of the drive in the PROFIBUS configuration.

Programm continuation: Switch control OFF - ON.

26106 Encoder %2 of axis %1 not found

Parameters: %1 = Axis name, spindle number

%2 = Encoder number

Explanation: For PROFIdrive only:

The encoder configured for the specified axis could not be found. For example, a PROFIBUS slave was configured on the NC but it is not contained in the SDB or defective hardware was reported for it.

Reaction: Mode group not ready.

The NC switches to follow-up mode.

Channel not ready.

NC Start disable in this channel.

Interface signals are set.

Alarm display.

NC Stop on alarm.

Remedy: Possible causes:

- MD 30240 $MA_ENC_TYPE not equal to 0 as a result of an oversight; the encoder should actually be simulated (= 0).
- MD 30220 $MA_ENC_MODULE_NR entered incorrectly, i.e. the logical drive numbers were transposed and an invalid value is stored for this drive in MD 13050 $MN_DRIVE_LOGIC_ADDRESS (see next paragraph), or a drive number which does not exist on the bus was entered (check the number for slaves, for example).
- MD 13050 $MN_DRIVE_LOGIC_ADDRESS contains values which were not configured on the PROFIBUS (i.e. the values are not in SDB Type 2000) or different addresses were selected for the input and output slots of the drive in the PROFIBUS configuration.
- A fatal encoder error was detected during the selection of the encoder (encoder defective, removed), so that park status cannot be left (in such a case, this alarm is triggered instead of alarm 25000/25001 - please refer to them for other possible causes of the error).

Programm continuation: Switch control OFF - ON.

26110 Independent drive stop/retract triggered

Explanation: For SINAMICS only:
Informational alarm: At least in one axis, a "drive-autonomous shutdown or retraction" was triggered.

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: Deselect drive-autonomous stop or retration. acknowledge alarm with reset.

Programm 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

Explanation: One axis with ESR configuration and $AA_ESR_ENABLE[axis] = 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[axis] = 0 before setting axis to NEUTRAL.
Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.

Programm 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

Explanation: $AA_ESR_ENABLE[axis] = 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[axis] = 1 to neutral axes (apart from single axes).
Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.

Programm 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

Explanation: With $AA_ESR_ENABLE[axis] = 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.

Programm 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

Explanation: 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[axis] = 1 should not be set with neutral axes.
Alarm can be suppressed via MD11415 $MN_SUPPRESS_ALARM_MASK_2 bit 6 = 1.

Programm 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

Explanation: 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.

Programm 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

29033 [Channel %1:] Axis exchange of axis %2 not possible, PLC axis movement not yet completed

Parameters: %1 = Channel number
%2 = Axis

Explanation: A PLC axis has not yet reached its end position and cannot be returned to a channel or neutralized. This alarm should not occur when PLC data block FC18 is used.

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

Remedy: Wait until the axis has reached the end position or terminate the movement with delete distance to go.

Programm continuation: Clear alarm with the RESET key. Restart part program

5.3 Cycle alarms

To view detailed description about individual alarms directly on the PPU, proceed as follows:

1. Press this key on the PPU to enter the alarm operating area.

2. Select the desired alarm.

3. Press this key to open the online help for the selected alarm.

Note: You can further press this softkey in the current help screen to show a complete list of all SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms. In addition, you can also use the following softkey to search for a specific alarm by number in this list:

4. Pressing this softkey exits the help system.

61000 **%[[Channel %1:] Block %2: %]No tool offset active**

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

Explanation:

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

Remedy: D-correction must be programmed before the cycle call.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61001 **[Channel %1:] Block %2: Thread pitch incorrectly defined**

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

Explanation:

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

Remedy: Check parameter for the thread size or setting for the pitch (contradict each other).

Programm continuation: Clear alarm with the RESET key. Restart part program

61002 %[[Channel %1:] Block %2: %]Type of machining incorrectly defined

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

Explanation:

Remedy: Modify VARI parameter.

Programm continuation: Clear alarm with the RESET key. Restart part program

61003 %[[Channel %1:] Block %2: %]No feed programmed in cycle

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

Explanation:

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

Remedy: Modify feed parameter.

Programm continuation: Clear alarm with the RESET key. Restart part program

61004 [Channel %1:] Block %2: Incorrect configuration of geometry axes

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

Explanation:

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61005 [Channel %1:] Block %2: 3rd geometry axis not available

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

Explanation: With an application on the turning machine with no Y-axis in the G18 plane.

Remedy: Check parameter on cycle call.

61006 %[[Channel %1:] Block %2: %]Tool radius too large

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

Explanation: The tool radius is too large for machining.

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

Remedy: Select a smaller tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61007 [Channel %1:] Block %2: Tool radius too small

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

Explanation: The tool radius is too small for machining.

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

Remedy: Select a larger tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61008 [Channel %1:] Block %2: No tool active

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

Explanation: Alarm triggered by following cycles:

Remedy: Select a tool.

61009 [Channel %1:] Block %2: Active tool number = 0

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

Explanation: No tool (T) has been programmed before the cycle call.

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

Remedy: Program tool (T).

Programm continuation: Clear alarm with the RESET key. Restart part program

61010 [Channel %1:] Block %2: Finishing allowance too large

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

Explanation: Causes of error:
- The finishing allowance for the base is greater than the total depth.
- The finishing allowance at the edge is greater than or equal to the tool diameter.

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

Remedy: Reduce finishing allowance.

Programm continuation: Clear alarm with the RESET key. Restart part program

61011 [Channel %1:] Block %2: Scaling not permissible

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

Explanation: A scale factor is active which is illegal for this cycle.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Modify scale factor.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61012 %[[Channel %1:] Block %2: %]Different scaling in planes

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61013 [Channel %1:] Block %2: Basic settings were changed, program cannot be executed

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation: The basic settings are not compatible with the generated program.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Check and, if necessary, change the basic settings.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61014 [Channel %1:] Block %2: Return plane exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Remedy: Check parameter RTP.

61015 [Channel %1:] Block %2: Contour is not defined

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61016 [Channel %1:] Block %2: System frame for cycles missing

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

Explanation:

Remedy: Set MD 28082: MM_SYSTEM_FRAME_MASK, Bit 5=1.

61017 %[[Channel %1:] Block %2: %]Function %4 not present in NCK

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61018 [Channel %1:] Block %2: function %4 not executable with NCK

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61019 %[[Channel %1:] Block %2: %]Parameter %4 incorrectly defined

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

Explanation: Example:
61019 parameter (S_MVAR: dec4) defined incorrectly
The value of the 4th decimal place (dec4 -> THOUSANDS digit) of the transfer parameter S_MVAR is defined incorrectly

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

Remedy: Check the value of the parameter.
If error message 61019 comes from CYCLE832, then:
61019 parameter S_TOLM: xx is incorrectly defined
1. Parameter S_TOLM lies outside the value range: S_TOLM UNITS digit 0 to 3. TENS digit 0 or 1
2. Parameter S_TOLM>3 and parameter S_OTOL not programmed and setting data SD55220 $SCS_FUNCTION_MASK_MILL_TOL_SET bit0=0
Remedy:
Set parameter S_TOLM in valid range (0..13), or,
Set the compatibility bit in SD55220 bit0=1 with the field technology when CYCLE832 is called (up to SW 2.6).
If the error message comes from CUST_832:
The manufacturer cycle CUST_832 must be upgraded to the latest cycle version.
If the error message comes from CYCLE9960:
1 (E996) : Compile Cycle E996 is not set
2 (E996) : Too many E996m_ files, only up to _E996m_10
3 E996 MD62736: Compensation limit exceeded
4 REF HEAD TRAFOTYP? Reference head only with swivel head or mixed kinematics
5 REF HEAD S_KNUM? Parameter S_KNUM 9000 = active WO or <> 1(G54)..99(G599)
6 REF HEAD? No head measured as reference head _OVR[105] <=0 _OVR[105]= transformation number
7 REF Work offset? _OVR[104] -> WO was measured with reference head
8 MVAR_DEC5=4/5? MVAR for E996 with only one RA incorrectly parameterized _DEC5=4 1.RA / _DEC5=5 2.RA
9 MD62738?? $MC_E996_FILE_LOCATION incorrectly parameterized
10 $NT_CNTRL bit7 <>0? with reference head $NT_CNTRL bit7=0 -> vector chain open
11 MODULO E996 RA is a modulo axis. The measuring range contains $MA_MODULO_RANGE_START, $MA_MODULO_RANGE_START must be a measuring position
12 MODULO E996 RA is a modulo axis. The measuring range contains $MA_MODULO_RANGE_START. The angle between the measuring points must be a divisor of $MA_MODULO_RANGE
14 MD11160 Authorization inadequate to write the kinematic chain
15 E996_FILE The active compensation file is not compatible with the active transformation.

Programm continuation: Clear alarm with the RESET key. Restart part program

61020 [Channel %1:] Block %2: Machining not possible with active TRANSMIT/TRACYL

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61021 [Channel %1:] Block %2: Parameter %4 value too high

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

Explanation:

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

Remedy: For measuring cycles:
- Correct the parameters displayed on the measuring cycle screen or measuring cycle interface.
-For CYCLE63 and trochoidal milling:
- The angle of contact CMAX must be less than or equal to 80° during roughing

Programm continuation: Clear alarm with the RESET key. Restart part program

61022 [Channel %1:] Block %2: Parameter %4 value too low

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

Explanation:

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

Remedy: - For CYCLE63 and trochoidal milling:
  - The angle of contact CMIN must be greater than 10° during roughing

Programm continuation: Clear alarm with the RESET key. Restart part program

61023 [Channel %1:] Block %2: Parameter %4 value must be unequal to zero

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61024 [Channel %1:] Block %2: Parameter %4 check value

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

Explanation:

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

Remedy: If the error message comes from CYCLE9960:
Start/End RA1: Set measuring range is not compatible with the traversing range (MD36100) of the rotary axis.
Start/End RA2

Programm continuation: Clear alarm with the RESET key. Restart part program

61025 [Channel %1:] Block %2: Check tool carrier position

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61026 [Channel %1:] Block %2: Cycle cannot be executed with NC function %4.

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61027 %[[Channel %1:] Block %2: %]Subroutine %4 does not exist

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

Explanation:

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

Remedy: - Check CYCLE62 call
- Check whether the subroutines specified on CYCLE62 call exist in the program storage

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61028 [Channel %1:] Block %2: Contour name %4 too long

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

Explanation:

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

Remedy: Error message in connection with contour call:
- Use shorter contour name

Error message in connection with contour turning:
-String lengths of the paths of the machined part contour or blank contour (if blank contour) or the updated blank contour (if residual machining) too long
-Use shorter directory or contour name

Error message in connection with contour milling:
-String lengths of the paths of the pocket/blank contour or the island contours too long
-Use shorter directory or contour name

Programm continuation: Clear alarm with the RESET key. Restart part program

61029 %[[Channel %1:] Block %2: %]Program name %4 too long

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

Explanation:

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

Remedy: - use shorter program name

Programm continuation: Clear alarm with the RESET key. Restart part program

61030 [Channel %1:] Block %2: Path not permitted: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61031 [Channel %1:] Block %2: Path not found: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61032 [Channel %1:] Block %2: File not found: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61033 [Channel %1:] Block %2: Incorrect file type: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61034 [Channel %1:] Block %2: File is full: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61035 [Channel %1:] Block %2: File in use: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61036 [Channel %1:] Block %2: NC storage limit reached: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61037 [Channel %1:] Block %2: No access rights to file: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61038 [Channel %1:] Block %2: Other file error: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61039 [Channel %1:] Block %2: Line not available: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61040 [Channel %1:] Block %2: Line longer than result variable: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61041 [Channel %1:] Block %2: Line range too large: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61042 [Channel %1:] Block %2: Program name %4 illegal

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

Explanation:

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

Remedy: In the case of multi-channel systems, the main program name must not end with _Cxx (xx stands for figures).
Rename main program.

Programm continuation: Clear alarm with the RESET key. Restart part program

61043 [Channel %1:] Block %2: Error affecting coordinate conversion (%4)

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

Explanation:

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

Remedy: 1: Type not specified
2: Error during tool identification
3: Meas. point 1 not available
4: Meas. point 2 not available
5: Meas. point 3 not available
4: Meas. point 4 not available
7: No reference point available
8: No direction of approach
9: Meas. points are identical
10: Alpha is incorrect
11: Phi is incorrect
12: Incorrect direction of approach
13: Lines do not intersect
14: Planes not available
15: No frame or incorrect frame selected
16: Not enough memory available
17: Internal error

Programm continuation: Clear alarm with the RESET key. Restart part program

61044 %[[Channel %1:] Block %2: %]Illegal character(s) in file name: %4

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

Explanation:

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

Remedy: Remove illegal character from file name
Permitted characters are: letters, figures, underscore, slash with path name

Programm continuation: Clear alarm with the RESET key. Restart part program

61045 [Channel %1:] Block %2: Job list not found: %4

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

Explanation:

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

Remedy: The specified job list could not be found.
Check name and contents of job list.
The job list must be in the same workpiece as the part program.

Programm continuation: Clear alarm with the RESET key. Restart part program

61046 [Channel %1:] Block %2: Part program not found in job list: %4

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

Explanation:

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

Remedy: Part program (main program) not found in specified job list in respective channel.
Check name and contents of job list.

Programm continuation: Clear alarm with the RESET key. Restart part program

61047 [Channel %1:] Block %2: Label name %4 too long

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

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - select shorter label name
Programm continuation: Clear alarm with the RESET key. Restart part program

61048 [Channel %1:] Block %2: Multi-channel data not found in job list: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Multi-channel data not found in job list.
Correct job list.
Programm continuation: Clear alarm with the RESET key. Restart part program

61049 [Channel %1:] Block %2: 1st spindle not programmed

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program 1st spindle in the screen.
Programm continuation: Clear alarm with the RESET key. Restart part program

61050 [Channel %1:] Block %2: Spindle programmed twice

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The same spindle was programmed twice.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Leave 2nd spindle empty or program other spindle.
Programm continuation: Clear alarm with the RESET key. Restart part program

61051 %[[Channel %1:] Block %2: %]Program name assigned twice

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The same program name was assigned twice.

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

Remedy: When using CYCLE952, the name of the main program must not be the same as the name of the cutting file (PRG) or the name of the updated blank contour (CONR).
When using CYCLE63 and/or CYCLE64, the name of the main program must not be the same as the name of the program to be generated (PRG).

Programm continuation: Clear alarm with the RESET key. Restart part program

61052 [Channel %1:] Block %2: Maximum spindle speed for main spindle not entered

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

Explanation: Max. speed for main spindle was not entered.

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

Remedy: Enter speed limit in program header or under settings.

Programm continuation: Clear alarm with the RESET key. Restart part program

61053 [Channel %1:] Block %2: Maximum spindle speed not entered for counterspindle

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

Explanation: Maximum speed of counterspindle was not entered.

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

Remedy: Enter speed limit in program header or under settings.

Programm continuation: Clear alarm with the RESET key. Restart part program

61054 [Channel %1:] Block %2: Programs started from various job lists: %4

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

Explanation: Programs from various job lists were started simultaneously.
This is illegal. All programs must be assigned to the same job list.

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

Remedy: Select desired job list again, and start programs afresh.

Programm continuation: Clear alarm with the RESET key. Restart part program

61055 [channel %1:] block %2: magazine location number too low: %4

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

Explanation: The magazine location number entered was too low.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61056 [channel %1:] block %2: magazine location number too high: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The magazine location number entered was too high.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61057 [Channel %1:] Block %2: magazine location number is no integer: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The magazine location number must be an integer.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61058 %[[channel %1:] block %2: %]Function %4 not released

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - CYCLE952: balance cutting function must be released via MD52218 $MCS_FUNCTION_MASK_TURN, bit6.
Programm continuation: Clear alarm with the RESET key. Restart part program

61059 %[[Channel %1:] Block %2: %]Tool preselection failed

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

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61060 %[[Channel %1:] Block %2: %]Function requires tool management

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

Explanation: --

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61061 %[[Channel %1:] Block %2: %]Different scaling in the plane and depth

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

Explanation:

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61062 %[[Channel %1:] Block %2: %]Axis position %4incorrectly programmed

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

Explanation:

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

Remedy: Check the last programmed axis position

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61063 %[[Channel %1:] Block %2: %]Tool at magazine location %4 is not a multitool

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

Explanation: A magazine location and a multitool location were programmed. However, there is no multitool at the magazine location.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Only program the magazine location (without multitool location).
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61064 %[[Channel %1:] Block %2: %]Invalid multitool location: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: A multitool location was programmed, at which there is no multitool.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program a valid multitool location.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61065 [Channel %1:] Block %2: A channel specified in the job list does not exist: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: A non-existent channel was specified in the job list.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Correct job list
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61066 %[[Channel %1:] Block %2: %]Cycle %4 requires G code G70 or G71

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Cycle requires G code G70 or G71
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program G70 or G71
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61067 [Channel %1:] Block %2:Tool spindle is in spindle mode: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: To activate the turning transformation, the tool spindle must be in axis mode. But it is in spindle mode.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Before activating the turning transformation, put the tool spindle in axis mode (with SPOS or M70).
Programm continuation: Clear alarm with the RESET key. Restart part program

61068 [Channel %1:] Block %2: Cycle requires G code %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: An incorrect G code is active when the cycle is called.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Activate the stated G code before calling the cycle. Multiple G codes may be presented for selection.
Example: Cycle requires G code: G70/G71
Programm continuation: Clear alarm with the RESET key. Restart part program

61069 [Channel %1:] Block %2: Axis not referenced %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: A referenced axis is required in the cycle, but an unreferenced axis was detected.
If possible, the axis identifier is stated in the alarm text.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Reference axis before calling the cycle.
Programm continuation: Clear alarm with the RESET key. Restart part program

61070 [Channel %1:] Block %2: Incorrect tool type for "shank" selection

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: For the "shank" selection, the tool must have an acute angle.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use "tip" selection.
Programm continuation: Clear alarm with the RESET key. Restart part program

61071 %[[Channel %1:] Block %2: %]This type of contour call is only possible with EES

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

Explanation: When a part program is called via EXTCALL without EES, the contour can only be called via "Contour name" or "Labels". This means that contour calls via "Subroutine" or "Labels in subroutine" are only possible if EES is active.

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

Remedy: - Program contour call via "Contour name" or "Labels"

Programm continuation: Clear alarm with the RESET key. Restart part program

61098 [Channel %1:] Block %2: %4

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

Explanation: This alarm is used for various purposes.
Please refer to the alarm text.

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

Remedy: Depends on alarm text

Programm continuation: Clear alarm with the RESET key. Restart part program

61099 [Channel %1:] Block %2: Internal cycle error (%4)

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61101 %[[Channel %1:] Block %2: %]Reference point defined incorrectly

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

Explanation:

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

Remedy: With an incremental specification of the depth, either different values have to be selected for the reference point (reference plane) and the retraction plane, or an absolute value must be specified for the depth.
When machining in the "manual machine" area, the following setting data must be checked and adapted to the machining.
SD 55260 $SCS_MAJOG_SAFETY_CLEARANCE (value greater than zero)
SD 55261 $SCS_MAJOG_RELEASE_PLANE

Programm continuation: Clear alarm with the RESET key. Restart part program

61102 %[[Channel %1:] Block %2: %]No spindle direction programmed

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

Explanation:

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

Remedy: Parameter SDIR (or SDR in CYCLE840) must be programmed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61103 [Channel %1:] Block %2: Number of holes is zero

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

Explanation:

Remedy: Check parameter NUM.

Programm continuation: Clear alarm with the RESET key. Restart part program

61104 [Channel %1:] Block %2: Contour violation of slots

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

Explanation: Incorrect parameterization of the milling pattern in the parameters which define the position of the slots/elongated holes on the circle and their form.

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61105 [Channel %1:] Block %2: Milling cutter radius too large

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

Explanation: The diameter of the cutter used is too large for the form to be machined.

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

Remedy: Either a tool with a smaller radius has to be used or the contour must be modified.

Programm continuation: Clear alarm with the RESET key. Restart part program

61106 [Channel %1:] Block %2: Number of or distance between circular elements

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

Explanation: Incorrect parameterization of NUM or INDA. The layout of the circle elements within a full circle is not possible.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Correct parameterization.
Programm continuation: Clear alarm with the RESET key. Restart part program

61107 [Channel %1:] Block %2: First drilling depth incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Modify drilling depth.
Programm continuation: Clear alarm with the RESET key. Restart part program

61108 [Channel %1:] Block %2: Illegal values for radius and insertion depth parameters

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The radius (_RAD1) and insertion depth (_DP1) parameters for defining the helix path for the depth infeed have been incorrectly specified.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Modify parameter.
Programm continuation: Clear alarm with the RESET key. Restart part program

61109 %[[Channel %1:] Block %2: %]Milling direction parameter incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The value of the parameter for the cutting direction _CDIR has been incorrectly defined.
Remedy: - Change milling direction.
- During pocket machining (CYCLE63), the selected milling direction must match the milling direction of centering/rough drilling.
Programm continuation: Clear alarm with the RESET key. Restart part program

61110 [Channel %1:] Block %2: Finishing allowance at the base is greater than the depth infeed

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The finishing allowance at the base has been specified greater than the maximum depth infeed.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Either reduce finishing allowance or increase depth infeed.
Programm continuation: Clear alarm with the RESET key. Restart part program

61111 [Channel %1:] Block %2: Infeed width is greater than the tool diameter

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The programmed infeed width is greater than the diameter of the active tool.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Infeed width must be reduced.
Programm continuation: Clear alarm with the RESET key. Restart part program

61112 [Channel %1:] Block %2: Tool radius negative

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The radius of the active tool is negative. This is illegal.
Remedy: Change the tool radius.
Programm continuation: Clear alarm with the RESET key. Restart part program

61113 [Channel %1:] Block %2: Parameter for corner radius too large

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The parameter for the corner radius _CRAD has been specified too large.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Reduce corner radius
Programm continuation: Clear alarm with the RESET key. Restart part program

61114 [Channel %1:] Block %2: Machining direction G41/G42 incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The machining direction of the cutter radius compensation G41/G42 has been incorrectly selected.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Change machining direction.
Programm continuation: Clear alarm with the RESET key. Restart part program

61115 [Channel %1:] Block %2: Approach or retract mode(straight / circle / plane / space) incorrectly defined

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

Explanation: The approach or retract mode to/from the contour has been incorrectly defined.

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

Remedy: Check parameter _AS1 or _AS2.

Programm continuation: Clear alarm with the RESET key. Restart part program

61116 [Channel %1:] Block %2: Approach or retract path = 0

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

Explanation: The approach or retract path has been specified with zero.

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

Remedy: Check parameter _LP1 or _LP2.

Programm continuation: Clear alarm with the RESET key. Restart part program

61117 %[[Channel %1:] Block %2: %]Active tool radius is smaller than or equal to zero

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

Explanation: The radius of the active tool is negative or zero.

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

Remedy: Modify radius.

Programm continuation: Clear alarm with the RESET key. Restart part program

61118 [Channel %1:] Block %2: Length or width = 0

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

Explanation: The length or width of the milling area is illegal.

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

Remedy: Check parameters _LENG and _WID.

Programm continuation: Clear alarm with the RESET key. Restart part program

61119 [Channel %1:] Block %2: Nominal or core diameter programmed incorrectly

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

Explanation: The nominal or core diameter was incorrectly programmed.

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

Remedy: Check thread geometry.

Programm continuation: Clear alarm with the RESET key. Restart part program

61120 [Channel %1:] Block %2: Thread type inside / outside not defined

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

Explanation: The thread type (internal/external) was not defined.

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

Remedy: The internal/external thread type must be entered.

Programm continuation: Clear alarm with the RESET key. Restart part program

61121 [Channel %1:] Block %2: Number of teeth per cutting edge is missing

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

Explanation: No value was entered for the number of teeth per cutting edge.

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

Remedy: Enter the number of teeth/cutting edges for the active tool into the tool list.

Programm continuation: Clear alarm with the RESET key. Restart part program

61122 [Channel %1:] Block %2: Safety distance incorrectly defined in plane

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

Explanation: The safety clearance is negative or zero. This is not allowed.

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

Remedy: Define the safety clearance.

Programm continuation: Clear alarm with the RESET key. Restart part program

61123 [Channel %1:] Block %2: CYCLE72 cannot be simulated

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61124 [Channel %1:] Block %2: Infeed width is not programmed

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

Explanation:

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

Remedy: With active simulation without tool, a value for the infeed width _MIDA must always be programmed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61125 [Channel %1:] Block %2: Technology selection parameter incorrectly defined

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

Explanation:

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

Remedy: Check technology selection parameter (_TECHNO).

Programm continuation: Clear alarm with the RESET key. Restart part program

61126 [Channel %1:] Block %2: Thread length too short

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

Explanation:

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

Remedy: Program lower spindle speed or raise reference point (reference plane).

Programm continuation: Clear alarm with the RESET key. Restart part program

61127 [Channel %1:] Block %2: Wrong definition of tapping axis transformation ratio (machine data)

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

Explanation:

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

Remedy: Check machine data 31050 and 31060 in the appropriate gear stage of the drilling axis.

Programm continuation: Clear alarm with the RESET key. Restart part program

61128 [Channel %1:] Block %2: Insertion angle = 0 for insertion with oscillation or helix

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

Explanation:

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

Remedy: Check parameter _STA2.

Programm continuation: Clear alarm with the RESET key. Restart part program

61129 [Channel %1:] Block %2: perpendic. approach and retraction during contour milling only allowed with G40

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61130 [Channel %1:] Block %2: positions of parallel axes cannot be compensated. No workpiece reference agreed.

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61131 **[Channel %1:] Block %2: parameter _GEO incorrect, _GEO=%4**

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61132 **[Channel %1:] Block %2: Parallel axis parameters incorrect, check values for parallel axis parameters ABS/INK**

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61133 **[Channel %1:] Block %2: 3rd parallel axis parameter incorrect, check axis name or GUD _SCW_N[]**

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61134 **[Channel %1:] Block %2: Rotary axis parameters incorrect, check values for rotary axis parameters ABS/INK**

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61135 [Channel %1:] Block %2: incorrect parameter sequence for approaching target position: %4

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61136 [Channel %1:] Block %2: no 3rd geometry axis agreed in GUD _SCW_N[]

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61137 [Channel %1:] Block %2: swiveling and parallel axes cycle are mutually exclusive because of workpiece reference $P_WPFRAME

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61138 [Channel %1:] Block %2: parameter %4 incorrectly defined for tool monitoring in cycles

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61139 [Channel %1:] Block %2: error in function Tool monitoring in cycles

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61140 [Channel %1:] Block %2: Main spindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the main spindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61141 [Channel %1:] Block %2: C axis of the main spindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the C axis of the main spindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61142 [Channel %1:] Block %2: Counterspindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the counterspindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61143 [Channel %1:] Block %2: C axis of the counterspindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the C axis of the counterspindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61144 [Channel %1:] Block %2: Tool spindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the tool spindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61145 [Channel %1:] Block %2: Linear axis of the counterspindle is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the linear axis of the counterspindle.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

61146 [Channel %1:] Block %2: B axis is not set up correctly

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

Explanation:

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

Remedy: Check the set up of the B axis.
Check machine data 20070 $MC_AXCONF_MACHAX_USED and 20080 $MC_AXCONF_CHANAX_NAME_TAB,

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61147 [Channel %1:] Block %2: Transformation not active: %4

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

Explanation:

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

Remedy: The stated transformation is not active.
You have to activate the transformation before you can use it.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61148 [Channel %1:] Block %2: Swivel plane with active turning tool not possible

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

Explanation: Swiveling the plane is not possible when a turning tool is active.

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

Remedy: Load a milling tool before you swivel the plane.
The alarm can be suppressed using SD 55410 $SCS_MILL_SWIVEL_ALARM_MASK.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61149 [Channel %1:] Block %2: Positioning a milling tool with active turning tool is not possible

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

Explanation: Milling tools cannot be positioned if a turning tool is active.

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

Remedy: Load a milling tool, before you call positioning.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61150 [Channel %1:] Block %2: Tool cannot be aligned - error code: %4

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

Explanation:

Remedy: Error code:
A: Only new swivel plane permitted, see parameter _ST
B: Angular range >= 360 -> angular range of the B axis >= 360 degrees
C: Rotary axis vector V1 is not equal to 0 1 0 -> rotary axis vector V1 must rotate around Y
See setup of swivel CYCLE800
D: Rotary axis vector V2 -> rotary axis vector V2 must rotate around X (1 0 0) or Z (0 0 1)
See setup of swivel CYCLE800
E: WCS ROT Y > 90 -> active rotation of the Work (WCS) around Y is >90 degrees. -90 to +90 is permitted
F: If geometry axis Y is not available -> SD55221 set bit 5 = 1
G: Initial setting defines ($TC_CARR37] HUNDREDTHOUSANDS digit -Z or -X) -> SD55221 set bit 5 = 1
See setup of swivel CYCLE800
H: Align tool, but active plane not G18
I: 1st turning technology (MD52200=1) and no Y axis available -> mirroring not allowed
J: 1st turning technology (MD52200=1) and no Y axis available -> rotation around Y not possible
K: Grinding technology (MD52200=3) and align tool -> SD55221 set bit 5=1
L: Grinding technology (MD52200=3) and align tool -> set MD20186 = 1

61151 [Channel %1:] Block %2: Orientation of tool not possible - error code: %4

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

Explanation:

Remedy: Causes of error:
1st error code = A -> only additive swivel plane permitted, see parameter _ST

61152 [Channel %1:] Block %2: B axis kinematics (turning technology) either not or incorrectly set up in Start-up of swivel cycle - error code: %4

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

Explanation:

Remedy: Causes of error:
1st error code = A123 -> B axis not an automatic rotary axis under ShopTurn (123 corresponds to parameter _TCBA)
2nd error code = B123 -> B axis not activated in swivel cycle start-up (kinematics)
(123 corresponds to $TC_CARR37[n], n ... number of the swivel data record)

61153 [Channel %1:] Block %2: No 'Rotary axes direct' swivel mode possible - error code: %4

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

Explanation:

Remedy: Error code:
A -> No tool or no cutting edge (D1..) active
B -> Swivel "no" and swivel "direct", swivel plane "additive" not permitted
C -> Input value for rotary axis 1 not in the grid of the Hirth tooth system
D -> Input value for rotary axis 2 not in the grid of the Hirth tooth system
E -> Swivel "direct" programmed in automatic, but swivel cycle not started up ($TC_CARR37 UNITS digit < =2)
F -> ROT ? G5.. rotation in settable WO (G54..) active, not permitted
ROT ? SETFRAME rotation in basic reference active, not permitted
ROT ? CHBFRAME rotation in basic active, not permitted
G: WPFRAME ? swivel mode additive and translations in workpiece reference (WPFRAME) not permitted
H: X0,Y0,Z0 not equal to 0, swivel mode additive and translations not permitted before swivel
See parameter S_ST 1st decimal place
I: Turning machine and initial setting -X or -Z not possible in G18
J: Turning machine and initial setting -X or -Z and counterspindle mirrored around Z
K: Turning machine and initial setting -Z and G19 not allowed
Initial setting, see setup of swivel CYCLE800
Note on error codes F, G, H:
If the Work (WCS) is rotated before programming swivel direct, and the swivel plane is rotated and moved additively, alarm 61153 is reported with error code F, G or H in order to avoid an impermissible mixing of frame operations and direct machine positions.
Alternatively, the machining program can be reprogrammed with swivel axis for each axis and swivel plane new or additionally,
or with new swivel and swivel plane.

61154 %[[Channel %1:] Block %2: %]Final depth wrongly programmed

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

Explanation:

Remedy: Input of end depth possible only absolutely or incrementally

61155 [Channel %1:] Block %2: Unit for plane infeed wrongly programmed

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

Explanation:

Remedy: Unit for plane infeed possible only in mm or % of tool diameter

61156 [Channel %1:] Block %2: Depth calculation wrongly programmed

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

Explanation:

Remedy: Depth calculation possible only with or without SDIS

61157 %[[Channel %1:] Block %2: %]Reference point wrongly programmed

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

Explanation:

Remedy: - Check reference point in screenform, input only -X, centered or +X
- During calibration of the length of the infeed axis in JOG, two causes can cause this alarm:
1. The value of the reference workpiece was incorrectly programmed in the parameterization screenform.
2. The length of the workpiece probe in the infeed axis has been incorrectly entered in the tool data.

61158 %[[Channel %1:] Block %2: %]Machining plane wrongly programmed

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

Explanation:

Remedy: Check machining plane (G17, G18 or G19) in connection with the parameter _DMODE

61159 [Channel %1:] Block %2: Machining plane on cycle call differs from the one in the position pattern

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

Explanation:

Remedy: Adjust the machining plane on cycle call to the machining plane in the position pattern.

61160 [Channel %1:] Block %2: Residual material remains stationary, reduce plane infeed

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

Explanation:

Remedy: Reduce plane infeed or slot width, or use milling cutter with larger diameter

61161 [Channel %1:] Block %2: Centering diameter or tool parameter (diameter, tip angle) are incorrect

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

Explanation:

Remedy: - Diameter of centering with tip angle of active tool not possible
- Entered workpiece diameter, tool diameter or tip angle of tool incorrect
- Diameter of tool only has to be entered if centering is to be on workpiece diameter.

Programm continuation: Clear alarm with the RESET key. Restart part program

61162 [Channel %1:] Block %2: Tool parameter diameter or tip angle incorrect

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

Explanation:

Remedy: - The tool parameter diameter or tip angle must be greater than zero
- Tip angle must be less than 180°

Programm continuation: Clear alarm with the RESET key. Restart part program

61163 [Channel %1:] Block %2: Infeed width in the plane too large

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

Explanation:

Remedy: --

61164 [Channel %1:] Block %2: Transformation has incorrect type %4

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

Explanation:

Remedy: The transformation has the incorrect type. Correct the transformation set up.

61165 [Channel %1:] Block %2: Transformation set up incorrectly: %4

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

Explanation:

Remedy: The transformation has been set up incorrectly. Correct the transformation set up.

61166 [Channel %1:] Block %2: Check machine data: %4

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

Explanation:

Remedy: The machine data must be checked. Adapt the setting of the machine data.

61167 [Channel %1:] Block %2: Transformation not set up or not active: %4

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

Explanation:

Remedy: The specified transformation is not set up or not active. Set up or activate the transformation.

61168 [Channel %1:] Block %2: Incorrect machining plane: %4

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

Explanation:

Remedy: The machining plane is incorrect. Program correct machining plane.

61169 [Channel %1:] Block %2: Spindle incorrectly programmed

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

Explanation: An incorrect spindle was programmed.

Remedy: Change the spindle selection.

61170 [Channel %1:] Block %2: Incorrect block plane (%4)

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

Explanation: An invalid block plane was used.

Remedy: Use a valid block plane.
It is only permissible that the block planes are used in an ascending sequence.

61171 [Channel %1:] Block %2: Block plane used twice (%4)

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

Explanation: The same block plane was used twice.

Remedy: Only nest the block planes in an ascending sequence.

61172 [Channel %1:] Block %2: Spindle groups must not be nested

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

Explanation: You have used spindles in several nested groups.

Remedy: Only use one spindle in one group plane with nested groups.

61173 [Channel %1:] Block %2: Supplementary run-in code only possible with spindle

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

Explanation: The supplementary run-in code can only be used in one block with spindle.

Remedy: Use a block with spindle.

61174 [Channel %1:] Block %2: Not possible to align milling tool with active turning tool

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

Explanation: It is not possible to align milling tools with an active turning tool.

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

Remedy: Load a milling tool before you call alignment.

Programm continuation: Clear alarm with the RESET key. Restart part program

61175 [Channel %1:] Block %2: angle of aperture programmed too small

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

Explanation: The angle of aperture of the text (_DF) in the engraving cycle is too small. This means that the text for engraving does not fit in the specified angle.

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

Remedy: Enter a larger angle of aperture.

Programm continuation: Clear alarm with the RESET key. Restart part program

61176 [Channel %1:] Block %2: text length programmed too small

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

Explanation: The text length (_DF) in the engraving cycle is too short. This means that the text for engraving is longer than the specified text length.

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

Remedy: Enter longer text length

Programm continuation: Clear alarm with the RESET key. Restart part program

61177 [Channel %1:] Block %2: polar text length > 360 degrees

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

Explanation: In the engraving cycle, the polar text length must not exceed 360 degrees.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Enter shorter text length.
Programm continuation: Clear alarm with the RESET key. Restart part program

61178 [Channel %1:] Block %2: code page not present

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The specified code page is not supported by the cycle.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use code page 1252.
Programm continuation: Clear alarm with the RESET key. Restart part program

61179 [Channel %1:] Block %2: character does not exist, no.: %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Character number
Explanation: The character entered in the text for engraving cannot be milled.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Enter another character.
Programm continuation: Clear alarm with the RESET key. Restart part program

61180 [Channel %1:] Block %2: No name assigned to swivel data record

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Although there are several swivel data blocks, no unique names have been assigned.
Remedy: Assign unique name to swivel data block ($TC_CARR34[n]), if machine data 18088 $MN_MM_NUM_TOOL_CARRIER is >1.

61181 [Channel %1:] Block %2: NCK software version is insufficient for the Swivel function

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Swivelling is not possible with the current NCK software version.
Remedy: Upgrade NCK software to NCK 75.00 or higher.

61182 [Channel %1:] Block %2: Name of swivel data record unknown: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The specified name of the swivel data block is unknown.

Remedy: Check name of swivel data record $TC_CARR34[n].

61183 [Channel %1:] Block %2: Swivel CYCLE800: Retraction mode parameter lies outside value range: %4

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

Explanation: The value of the retraction mode parameter (_FR) lies outside the valid range.

Remedy: Swivel CYCLE800: Check transfer parameter _FR. Value range 0 to 8

61184 [Channel %1:] Block %2: No solution possible with current input angle values

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

Explanation: The surface defined via the input angle cannot be processed with the machine.

Remedy: -Check the angles entered for swiveling the machining plane: %4
-Parameter _MODE coding incorrect, e.g. rotation axis-wise YXY

61185 [Channel %1:] Block %2: Invalid angular ranges of rotary axes in swivel data record: %4

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

Explanation: The angular range of the rotary axes is invalid.
Check setup of the swivel CYCLE800.
Parameter $TC_CARR30[n] to $TC_CARR33[n] n number of swivel data record
Example: Rotary axis 1 modulo 360 degrees: -> $TC_CARR30[n]=0 $TC_CARR32[n]=360

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

Remedy: Check setup of swivel cycle CYCLE800.

Programm continuation: Clear alarm with the RESET key. Restart part program

61186 [Channel %1:] Block %2: Invalid rotary axis vectors --> Check setup of the swivel cycle CYCLE800.

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

Explanation: No or incorrect entry for rotary axis vector V1 or V2.

Remedy: Check commissioning of swivel CYCLE800
Check rotary axis vector V1xyz: $TC_CARR7[n], $TC_CARR8[n], $TC_CARR9[n]
Check rotary axis vector V2xyz: $TC_CARR10[n], $TC_CARR11[n], $TC_CARR12[n]
If 2nd rotary axis not available ($TC_CARR35[n]=""), then it is possible that V2xyz=0
n Number of the swivel data set

61187 [Channel %1:] Block %2: Check setup of the swivel cycle CYCLE800. - Error code: %4

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

Explanation:

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

Remedy: Error code: See current cycles software version notes in siemense.txt

Programm continuation: Clear alarm with the RESET key. Restart part program

61188 [Channel %1:] Block %2: No axis name agreed for rotary axis 1 -> Check setup of the swivel cycle CYCLE800.

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

Explanation: No axis name was specified for the rotary axis 1.

Remedy: Check setup of swivel CYCLE800.
Axis name of rotary axis 1 see parameter $TC_CARR35[n] n number of swivel data record

61189 [Channel %1:] Block %2: Swivel direct: Invalid rotary axis positions: %4

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

Explanation: Swivel direct: Check input values of rotary axes.

Remedy: Swivel mode direct: check the input values of the rotary axes or commissioning for swivel CYCLE800.
Check angular range of rotary axes in swivel data record n:
Rotary axis 1: $TC_CARR30[n], $TC_CARR32[n]
Rotary axis 2: $TC_CARR31[n], $TC_CARR33[n]
If values are entered in the work offset (WO) of the rotary axes, and machine data MD21186=0:
Value in WO of the rotary axis does not correspond to the angular range, rotary axis 1 or 2
Value in WO of the rotary axis plus input value does not correspond to the angular range, rotary axis 1 or 2
Note: for modulo axes, for swivel direct, the input value is calculated in the modulo range
Example: angular range, rotary axis modulo 0 up to 360 input value =-21 rotary axis moves to 339 degrees

61190 [Channel %1:] Block %2: Unable to retract prior to swiveling -> Error code: %4

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

Explanation:

Remedy: If tool carrier is active, check commissioning of swivel CYCLE800. Parameter $TC_CARR37[n] 7th and 8th decimal places n number of swivel data set
If align turning tool with milling/turning machine (transformation on basic kinematic chain) is active,
check parameter $NT_IDENT[n,0] 7th and 8th decimal places (n = name of transformation)
Error code:
A: Retraction Z not set up
B: Retraction Z XY not set up
C: Retraction in tool direction max. not set up
D: Retraction in tool direction incremental not set up
E: Retraction in tool direction: NC function CALCPOSI reports error
For the CALCPOSI function, the axes must be referenced. Check machine data MD20700.
F: Retraction in tool direction: no tool axis available
Missing geometry axis (tool axis) Z with G17, Y with G18 or X with G19
G: Retraction in tool direction max.: negative retraction path
H: Retraction in tool direction incremental: negative retraction path not permitted
I: Retraction commissioning swivel CYCLE800 not set up
J: Retraction in tool direction is not permitted, because current Work (WCS) is mirrored
Tool carrier is active
K: Retraction and swivel plane and transformation on basic kinematic chain are active
Retraction with transformation only if align turning tool is active
L: Retraction in tool direction and align tool are only permitted if SD42954 $SC_TOOL_ORI_CONST_M=0
Transformation on basic kinematic chain is active
M: Retraction in tool direction and align tool are only permitted if SD42956 $SC_TOOL_ORI_CONST_T=0
Transformation on basic kinematic chain is active
N: Retraction in tool direction and swivel plane toward counterspindle are not possible
Turning machine with active tool carrier and initial setting -X (see setup swivel)
O: Retraction in tool direction, align tool with counterspindle is not possible
P: Retraction in tool direction in G18 toward counterspindle is not possible
Q: Retraction in tool direction is not permitted because the current Work (WCS) is mirrored
Align turning tool with milling/turning machine (transformation on basic kinematic chain) active
R: Retraction in tool direction is not possible with align tool
Align turning tool with milling/turning machine (transformation on basic kinematic chain) active

61191 [Channel %1:] Block %2: Multi-axis transformation not set up. Error code: %4

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

Explanation:

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

Remedy: Error code:
Number or parameter name of multi-axis transformation

Programm continuation: Clear alarm with the RESET key. Restart part program

61192 [Channel %1:] Block %2: Additional multi-axis transformations not set up. Error code: %4

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

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Error code:
Number or parameter name of multi-axis transformation
Programm continuation: Clear alarm with the RESET key. Restart part program

61193 [Channel %1:] Block %2: compressor option not set up

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61194 [Channel %1:] Block %2: spline interpolation option not set up

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61195 [Channel %1:] Block %2: Aligning a turning tool is only possible with active turning tool

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Aligning turning tools is only possible with one active turning tool.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Change a turning tool, before you call alignment.
Programm continuation: Clear alarm with the RESET key. Restart part program

61196 [Channel %1:] Block %2: No swiveling in JOG --> Multi-axis transformations and TCARR activated at the same time

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Multi-axis transformations (TRAORI) and Toolcarrier (TCARR) activated at the same time.

Remedy: Deselection of multi-axis transformation with TRAFOOF
or deselection of Toolcarrier (TCARR) with CYCLE800()

61197 [Channel %1:] Block %2: Swiveling - plane not allowed -> Error code %4

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

Explanation:

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

Remedy: Error code:

Programm continuation: Clear alarm with the RESET key. Restart part program

61198 [Channel %1:] Block %2: Swiveling with kinematic chain -> Error code: %4

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

Explanation:

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

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61199 [Channel %1:] Block %2: Swiveling - tool not allowed -> Error code: %4

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

Explanation: Error is also output if no swivel data record is active and only tool approach is programmed.

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

Remedy: A CYCLE800 must be called with a valid swivel data record before orientation.
Error code:
A: Orientation of tool and change of swivel data record not allowed

Programm continuation: Clear alarm with the RESET key. Restart part program

61200 [Channel %1:] Block %2: Too many elements in machining block

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

Explanation: The machining block contains too many elements.

Remedy: Check the machining block, delete some elements if required.

61201 [Channel %1:] Block %2: Wrong sequence in machining block

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

Explanation: The sequence of elements in the machining block is invalid.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Sort the sequence in the machining block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61202 [Channel %1:] Block %2: No technology cycle

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No technology cycle was programmed in the machining block.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program a technology block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61203 [Channel %1:] Block %2: No position cycle

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No positioning cycle was programmed in the machining block.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program positioning block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61204 [Channel %1:] Block %2: Technology cycle unknown

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The specified technology cycle in the machining block is unknown.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Delete and reprogram the technology block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61205 [Channel %1:] Block %2: Position cycle unknown

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The specified positioning cycle in the machining block is unknown.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Delete and reprogram the positioning block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61206 [Channel %1:] Block %2: Synchronizing possible only when using a job list

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Synchronizing with a counterspindle step in another channel is possible only if a job list is used.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Create job list and add programs for the individual channels.
Programm continuation: Clear alarm with the RESET key. Restart part program

61207 [Channel %1:] Block %2: No counterspindle step found for synchronizing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No counterspindle step was found in any channel with which this channel could synchronize.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Control program.
Delete step for synchronizing if it is not required.
Programm continuation: Clear alarm with the RESET key. Restart part program

61208 [Channel %1:] Block %2: Assign parameters for main spindle chuck in the spindle chuck data

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The parameters for the main spindle chuck in the spindle chuck data are not assigned.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Indicate parameters ZCn, ZSn and ZEn in the screen "Parameters" > "Setting data" > "Spindle chuck data".
Programm continuation: Clear alarm with the RESET key. Restart part program

61209 [channel %1:] block %2: counterspindle step programmed in several channels

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Counterspindle step may be programmed in one channel only.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: The step "Counterspindle: synchronizing" must be used in the other channels.
Programm continuation: Clear alarm with the RESET key. Restart part program

61210 [Channel %1:] Block %2: Block search element not found

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The element specified for the block search does not exisit.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Repeat block search.
Programm continuation: Clear alarm with the RESET key. Restart part program

61211 [Channel %1:] Block %2: Absolute reference missing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: An incremental indication was made, but the absolute reference is unknown.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Program an absolute position prior to using incremental indications.
Programm continuation: Clear alarm with the RESET key. Restart part program

61212 %[[Channel %1:] Block %2: %]Wrong tool type

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The tool type is not suitable for machining.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select a new tool type.
Programm continuation: Clear alarm with the RESET key. Restart part program

61213 [Channel %1:] Block %2: Circle radius too small

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The programmed circle radius is too small.
Remedy: Correct the circle radius, center point or end point.

61214 [Channel %1:] Block %2: No pitch programmed

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

Explanation: No thread/helical pitch has been entered.

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

Remedy: Program a pitch.

Programm continuation: Clear alarm with the RESET key. Restart part program

61215 [Channel %1:] Block %2: Unfinished dimension incorrectly programmed

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

Explanation: Check the blank spigot dimensions. The blank spigot must be larger than the production part spigot.

Remedy: Check parameters _AP1 and _AP2.

61216 %[[Channel %1:] Block %2: %]Feed/tooth only possible with cutting tools

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

Explanation: Feed per tooth is only possible with milling tools.

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

Remedy: As alternative, set a different feed type.

Programm continuation: Clear alarm with the RESET key. Restart part program

61217 [Channel %1:] Block %2: Cutting speed programmed for tool radius 0

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

Explanation: To be able to work with cutting speed, the tool radius has to be specified.

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

Remedy: Enter a value for cutting speed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61218 [Channel %1:] Block %2: Feed/tooth programmed, but number of tools equals zero

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

Explanation: For feed per tooth, the number of teeth has to be specified.

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

Remedy: Enter the number of teeth on the milling tool in the "Tool list" menu.
Programm continuation: Clear alarm with the RESET key. Restart part program

61219 [Channel %1:] Block %2: Tool radius too large

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The tool radius is too large for machining.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select a suitable tool.
Programm continuation: Clear alarm with the RESET key. Restart part program

61220 [Channel %1:] Block %2: Tool radius too small

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The tool radius is too small for machining.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select a suitable tool.
Programm continuation: Clear alarm with the RESET key. Restart part program

61221 [Channel %1:] Block %2: No tool active

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No tool active.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select a suitable tool.
Programm continuation: Clear alarm with the RESET key. Restart part program

61222 [Channel %1:] Block %2: Plane infeed greater than tool diameter

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The plane infeed must not be greater than the tool diameter.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Reduce plane infeed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61223 [Channel %1:] Block %2: Approach path too small

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

Explanation: The approach path must not be less than zero.

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

Remedy: Enter a greater value for the approach path.

Programm continuation: Clear alarm with the RESET key. Restart part program

61224 [Channel %1:] Block %2: Retract path too small

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

Explanation: The retract path must not be less than zero.

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

Remedy: Enter a greater value for the retract path.

Programm continuation: Clear alarm with the RESET key. Restart part program

61225 [Channel %1:] Block %2: Swivel data record unknown

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

Explanation: An attempt was made to access a swivel data block which has not been defined.

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

Remedy: Select another swivel data block or define a new swivel data block.

Programm continuation: Clear alarm with the RESET key. Restart part program

61226 [Channel %1:] Block %2: Swivel head cannot be exchanged

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

Explanation: The parameter "Swivel data block" is set to "No". In spite of this, an attempt has been made to change the swivel head.

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

Remedy: Set the parameter "Swivel data block" in the setup screen "Rotary axes" to "Automatic" or "Manual".

Programm continuation: Clear alarm with the RESET key. Restart part program

61227 [Channel %1:] Block %2: Target position cannot be reached: %4

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

Explanation: The target position of the approach motion is outside the software limit switch. This situation may arise from swiveling or coordinate rotations.

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

Remedy: If possible, lower retraction plane. Select alternative solution for swiveling (direction +/-) or clamp on workpiece differently.

Programm continuation: Clear alarm with the RESET key. Restart part program

61228 [Channel %1:] Block %2: Retraction plane for swiveling with swivel head not reached due to software limit switches

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

Explanation: Retraction plane not reached!

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

Remedy: Correct retraction plane

Programm continuation: Clear alarm with the RESET key. Restart part program

61229 [Channel %1:] Block %2: The external retraction plane must be greater than the internal retraction plane

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

Explanation: The external retraction plane must be greater than the internal retraction plane.

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

Remedy: Correct the retraction planes.

Programm continuation: Clear alarm with the RESET key. Restart part program

61230 [Channel %1:] Block %2: Tool probe diameter too small

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

Explanation: The tool probe has not been calibrated correctly.

Remedy: 840D sl - up to SW 1.x:
Check the following variables in data block GUD7: E_MESS_MT_DR[n] or E_MESS_MT_DL[n] for probe n+1
840D sl/828D - as from SW 2.5:
Check the following machine or setting data: 51780 $MNS_J_MEA_T_PROBE_DIAM_RAD[n]

61231 [Channel %1:] Block %2: ShopMill program %4 not executable, as not tested by ShopMill

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

Explanation: Before a ShopMill program can be executed, it has to be tested by ShopMill.

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

Remedy: The program first has to be simulated in ShopMill or loaded into the operating mode "Machine auto" by ShopMill.

Programm continuation: Clear alarm with the RESET key. Restart part program

61232 [Channel %1:] Block %2: Impossible to load magazine tool

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

Explanation: Only manual tools may be loaded into a swivel head in which only manual tools can be loaded.

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

Remedy: Load a manual tool into the swivel head or set the parameter "Tool change" on the setup screenform "Rotary axes" to "Automatic".

Programm continuation: Clear alarm with the RESET key. Restart part program

61233 [Channel %1:] Block %2: Thread angle wrongly defined

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

Explanation: The thread angles were specified too large or too small.

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

Remedy: Check thread geometry.

Programm continuation: Clear alarm with the RESET key. Restart part program

61234 [Channel %1:] Block %2: ShopMill subroutine %4 cannot be executed, as not tested by ShopMill

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Subroutine name

Explanation: Before a ShopMill subroutine can be used, it has to be tested by ShopMill.

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

Remedy: The subroutine first has to be simulated in ShopMill or loaded into the ShopMill operating mode "Machine auto".

Programm continuation: Clear alarm with the RESET key. Restart part program

61235 **[Channel %1:] Block %2: ShopTurn program %4 cannot be executed as not tested by ShopTurn.**

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

Explanation: Before a ShopTurn program can be executed, it has to be tested by ShopTurn.

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

Remedy: First simulate the program in ShopTurn or load it into the ShopTurn operating mode "Machine auto".

Programm continuation: Clear alarm with the RESET key. Restart part program

61236 **[Channel %1:] Block %2: ShopTurn subroutine %4 cannot be executed as not tested by ShopTurn.**

Parameters: %1 = Channel number
%2 = Block number, label
%4 = Subroutine name

Explanation: Before a ShopTurn subroutine can be used, it has to be tested by ShopTurn.

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

Remedy: Simulate the subroutine first in ShopTurn or load it into the ShopTurn operating mode "Machine auto".

Programm continuation: Clear alarm with the RESET key. Restart part program

61237 **[Channel %1:] Block %2: Retraction direction unknown. Withdraw tool manually!**

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

Explanation: The tool is in the retraction area and it is unknown in which direction it can be travelled out of it.

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

Remedy: Manually retract the tool from the retraction area defined in the program header and restart the program.

Programm continuation: Clear alarm with the RESET key. Restart part program

61238 **[Channel %1:] Block %2: Machining direction unknown!**

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

Explanation: The direction of the next machining is unknown.

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

Remedy: Please contact the responsible Siemens regional office.

Programm continuation: Clear alarm with the RESET key. Restart part program

61239 [Channel %1:] Block %2: Tool change point lies within retraction area!

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

Explanation: The tool change point has to be far enough outside the retraction area so that when the revolver is swiveled, no tool extends into the retraction area.

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

Remedy: Specify another tool change point.

Programm continuation: Clear alarm with the RESET key. Restart part program

61240 %[[Channel %1:] Block %2: %]Wrong feed type

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

Explanation: The feed type is not possible for this machining.

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

Remedy: Check feed type

Programm continuation: Clear alarm with the RESET key. Restart part program

61241 [Channel %1:] Block %2: Retraction plane not defined for this machining direction

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

Explanation: No retraction plane has been defined for the selected machining direction.

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

Remedy: Define the missing retraction plane.

Programm continuation: Clear alarm with the RESET key. Restart part program

61242 [Channel %1:] Block %2: Wrong machine direction

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

Explanation: The machining direction has been specified incorrectly.

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

Remedy: Check the programmed machining direction.

Programm continuation: Clear alarm with the RESET key. Restart part program

61243 [Channel %1:] Block %2: Correct tool change point, tool tip in retraction area!

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

Explanation: The tool change point must be situated so far outside the retraction area that no tool protrudes into the retraction area on turret swivelling.

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

Remedy: Specify another tool change point.

Programm continuation: Clear alarm with the RESET key. Restart part program

61244 [Channel %1:] Block %2: Pitch change leads to undefined thread

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

Explanation: The specified pitch change causes a reversal of the thread direction.

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

Remedy: Check thread pitch change and thread geometry.

Programm continuation: Clear alarm with the RESET key. Restart part program

61245 [Channel %1:] Block %2: Machining plane does not match modal machining plane

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

Explanation: Machining plane does not match modal one.

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

Remedy: Check the machining plane.

Programm continuation: Clear alarm with the RESET key. Restart part program

61246 [Channel %1:] Block %2: Safety distance too small

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

Explanation: The safety clearance is too small for machining.

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

Remedy: Increase safety clearance.

Programm continuation: Clear alarm with the RESET key. Restart part program

61247 [Channel %1:] Block %2: Blank radius too small

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

Explanation: The blank radius is too small for machining.

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

Remedy: Increase blank radius.

Programm continuation: Clear alarm with the RESET key. Restart part program

61248 [Channel %1:] Block %2: Infeed too small

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

Explanation: The infeed is too small for machining.

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

Remedy: Increase infeed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61249 [Channel %1:] Block %2: Number of edges too small

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

Explanation: The number of edges is too small.

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

Remedy: Increase number of edges.

Programm continuation: Clear alarm with the RESET key. Restart part program

61250 [Channel %1:] Block %2: Width across flats/edge length too small

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

Explanation: The width across flats/edge length is too small.

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

Remedy: Increase key width/edge length.

Programm continuation: Clear alarm with the RESET key. Restart part program

61251 [Channel %1:] Block %2: Width across flats/edge length too large

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

Explanation: The width across flats/edge length is too large.

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

Remedy: Decrease key width/edge length.

Programm continuation: Clear alarm with the RESET key. Restart part program

61252 [Channel %1:] Block %2: Chamfer/radius too large

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

Explanation: Chamfer/radius is too large.

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

Remedy: Decrease chamfer/radius.

Programm continuation: Clear alarm with the RESET key. Restart part program

61253 [Channel %1:] Block %2: No finishing allowance programmed

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

Explanation: No finishing allowance has been entered.

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

Remedy: Programm a finishing allowance.

Programm continuation: Clear alarm with the RESET key. Restart part program

61254 [Channel %1:] Block %2: Error while traveling to fixed stop

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

Explanation: Error on travelling to fixed stop.

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

Remedy: specify another Z1 position for gripping the counterspindle.

Programm continuation: Clear alarm with the RESET key. Restart part program

61255 [Channel %1:] Block %2: Error during cut-off: Tool broken?

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

Explanation: Cut-off could not be completed. A tool breakage might have occurred.

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

Remedy: Check the tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61256 [Channel %1:] Block %2: Mirroring not allowed at program start. Deselect work offset!

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

Explanation: Mirroring impermissible at program start.

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

Remedy: Deselect work offset.

Programm continuation: Clear alarm with the RESET key. Restart part program

61257 [Channel %1:] Block %2: incomplete setup of counterspindle

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

Explanation: Setup of the counterspindle is incomplete.

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

Remedy: The following machine and setting data must be set for the counterspindle:
- MD52206 $MCS_AXIS_USAGE
- SD55232 $SCS_SUB_SPINDLE_REL_POS
- SD55550 $SCS_TURN_FIXED_STOP_DIST
- SD55551 $SCS_TURN_FIXED_STOP_FEED
- SD55552 $SCS_TURN_FIXED_STOP_FORCE

Programm continuation: Clear alarm with the RESET key. Restart part program

61258 [Channel %1:] Block %2: Assign parameters for counterspindle chuck in the spindle chuck data

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

Explanation: The parameters for the counterspindle chuck in the spindle chuck data have not been assigned.

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

Remedy: Indicate parameters ZCn, ZSn and ZEn in the screen "Parameters" > "Setting data" > "Spindle chuck data".

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61259 [Channel %1:] Block %2: program contains new machining steps from ShopMill %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = ShopMill version

Explanation: The program has been created with a ShopMill version that is higher than the existing one.

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

Remedy: Delete the machining step and reprogram machining if required.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61260 [Channel %1:] Block %2: program contains new machining steps from ShopTurn %4

Parameters: %1 = Channel number
%2 = Block number, label
%4 = ShopTurn version

Explanation: The program has been created with a ShopTurn version that is higher than the existing one.

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

Remedy: Delete the machining step and reprogram machining if required.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61261 [Channel %1:] Block %2: center offset too large

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

Explanation: The center offset on center drilling is larger than permissible.

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

Remedy: Enter lower eccentricity ($SCS_DRILL_MID_MAX_ECCENT).

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61262 [Channel %1:] Block %2: pitch not possible with selected tool

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

Explanation: The pitch of the tap does not match the programmed pitch.

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

Remedy: Use a tap with the programmed pitch.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61263 [Channel %1:] Block %2: Chained ShopMill program blocks not permissible in subprogram on pos. pattern

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

Explanation: If a subroutine is called from a position pattern, the subroutine itself must not include a position pattern.

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

Remedy: Reprogram machining.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61264 [Channel %1:] Block %2: Chained ShopTurn program blocks not permissible in subprogram on pos. pattern

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

Explanation: If a subroutine is called from a position pattern, the subroutine itself must not include a position pattern.

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

Remedy: Reprogram machining.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61265 [Channel %1:] Block %2: Too many restrictions, use rectangular pocket

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

Explanation: In face milling a maximum of only 3 sides can be delimited.

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

Remedy: Use pocket cycle.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61266 [Channel %1:] Block %2: Illegal machining direction

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

Explanation: In face milling, the delimitations and the direction of machining do not match.

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

Remedy: Select another direction of machining.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61267 [Channel %1:] Block %2: Plane infeed too large, residual corners remain

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

Explanation: In face milling, the plane infeed must not exceed 85%.

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

Remedy: Select a smaller plane infeed, as otherwise residual corners will be left over.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61268 [Channel %1:] Block %2: Illegal machining direction, residual corners are left over.

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

Explanation: In face milling, the machining direction does not match the selected delimitations.

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

Remedy: The machining direction must be selected to match the delimitations.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61269 [Channel %1:] Block %2: External tool diameter too small

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

Explanation: Incorrect tool definition.

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

Remedy: Check angle and diameter of the tool used.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61270 %[[Channel %1:] Block %2: %]Chamfer width too small

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

Explanation: Chamfer width selected too small.

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

Remedy: Increase the chamfer width.

Programm continuation: Clear alarm with the RESET key. Restart part program

61271 %[[Channel %1:] Block %2: %]Chamfer width greater than tool radius

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

Explanation: Chamfer width larger than tool radius.

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

Remedy: Use a larger tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61272 %[[Channel %1:] Block %2: %]Insertion depth too small

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

Explanation: Insertion depth on chamfering too small.

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

Remedy: Increase the insertion depth.

Programm continuation: Clear alarm with the RESET key. Restart part program

61273 %[[Channel %1:] Block %2: %]Insertion depth too large

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

Explanation: Insertion depth on chamfering too large.

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

Remedy: Decrease the insertion depth.

Programm continuation: Clear alarm with the RESET key. Restart part program

61274 %[[Channel %1:] Block %2: %]Invalid tool angle

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

Explanation: Invalid tool angle.

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

Remedy: Check tool angle

Programm continuation: Clear alarm with the RESET key. Restart part program

61275 [Channel %1:] Block %2: Target point violates software limit switch!

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

Explanation: Due to a swivel action, the end point is outside the software limit switches.

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

Remedy: Select another retraction plane or approach a suitable interpolation point.

Programm continuation: Clear alarm with the RESET key. Restart part program

61276 [Channel %1:] Block %2: External tool diameter required for restrictions

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

Explanation: Outer tool diameter required in case of delimitations.

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

Remedy: Specify the outer tool diameter.

Programm continuation: Clear alarm with the RESET key. Restart part program

61277 [Channel %1:] Block %2: Tool diameter larger than restriction

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

Explanation: Tool diameter larger than delimitation.

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

Remedy: Use a smaller tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61278 [Channel %1:] Block %2: If tool angle is larger than 90°, both tool diameters must be equal

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

Explanation: For tool angles larger than 90°, the two tool diameters must be identical.

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

Remedy: Correct the tool angle or the tool diameters.

Programm continuation: Clear alarm with the RESET key. Restart part program

61279 [Channel %1:] Block %2: If tool angle equals 90°, both tool diameters must be equal

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

Explanation: For tool angles equal to 90°, the two tool diameters must be identical.

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

Remedy: Correct the tool angle or the tool diameters.

Programm continuation: Clear alarm with the RESET key. Restart part program

61280 [Channel %1:] Block %2: %4- Mirroring missing in work offset for counterspindle

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

Explanation: The work offset for counterspindle machining does not have Z mirroring.

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

Remedy: Select Z mirroring for the work offset used.

Programm continuation: Clear alarm with the RESET key. Restart part program

61281 [Channel %1:] Block %2: starting point of machining outside retraction planes

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

Explanation: The starting point of machining is outside the retraction planes.
It is derived from the programmed geometry plus the safety distance in the selected machining direction.
The calculated point must lie within the retraction area to ensure safe approach.

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

Remedy: Adjust the retraction planes.

Programm continuation: Clear alarm with the RESET key. Restart part program

61282 [Channel %1:] Block %2: end point of machining outside retraction planes

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

Explanation: The end point of machining is outside the retraction planes.
It is calculated internally (according to the selected machining strategy).

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

Remedy: Adjust the retraction planes.

Programm continuation: Clear alarm with the RESET key. Restart part program

61283 [Channel %1:] Block %2: direct approach not possible, as tool change required

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

Explanation: After block search a position is to be reached by direct approach, but a tool change is required before.

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

Remedy: First execute a manual tool change, then restart the block search.

Programm continuation: Clear alarm with the RESET key. Restart part program

61284 [Channel %1:] Block %2: starting point cannot be approached without collision. Pre-position tool manually

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

Explanation: The starting point cannot be approached without collisions.

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

Remedy: Preposition the tool manually.

Programm continuation: Clear alarm with the RESET key. Restart part program

61285 [Channel %1:] Block %2: parking position is below return plane XRA.

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

Explanation: The parking position is below retraction plane XRA.

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

Remedy: Move the parking position above retraction plane XRA.

Programm continuation: Clear alarm with the RESET key. Restart part program

61286 [Channel %1:] Block %2: machining not possible, check tool angle.

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

Explanation: Machining not possible with the specified tool.

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

Remedy: Use a suitable tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61287 %[[Channel %1:] Block %2: %]No master spindle active.

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

Explanation: No master spindle active.

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

Remedy: Activate the master spindle (machine data 20090).

Programm continuation: Clear alarm with the RESET key. Restart part program

61288 [Channel %1:] Block %2: Main spindle not set up

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

Explanation:

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

Remedy: Set up main spindle in MD52206 $MCS_AXIS_USAGE.

Programm continuation: Clear alarm with the RESET key. Restart part program

61289 [Channel %1:] Block %2: Counterspindle not set up

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

Explanation:

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

Remedy: Set up counter spindle in MD52206 $MCS_AXIS_USAGE.

Programm continuation: Clear alarm with the RESET key. Restart part program

61290 [Channel %1:] Block %2:Tool spindle not set up

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

Explanation:

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

Remedy: CYCLE210: Set up the tool spindle in MD52206 $MCS_AXIS_USAGE.
Check measuring cycles: MD35000 $MA_SPIND_ASSIGN_TO_MACHAX

Programm continuation: Clear alarm with the RESET key. Restart part program

61291 [Channel %1:] Block %2: Linear axis of counterspindle not set up

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

Explanation:

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

Remedy: Set up the linear axis of the counter spindle in MD52206 $MCS_AXIS_USAGE.

Programm continuation: Clear alarm with the RESET key. Restart part program

61292 [Channel %1:] Block %2: B axis not set up

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

Explanation:

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

Remedy: Set up the B axis in MD52206 $MCS_AXIS_USAGE.

Programm continuation: Clear alarm with the RESET key. Restart part program

61293 [Channel %1:] Block %2: Tool %4 has no spindle rotation direction

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

Explanation:

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

Remedy: Select spindle rotation direction in tool list.

Programm continuation: Clear alarm with the RESET key. Restart part program

61294 [Channel %1:] Block %2: Active radius/diameter setting does not match reset setting

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

Explanation:

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

Remedy: Before starting up the program, set G group 29 (DIAMON, DIAMOF etc.) to reflect the corresponding reset value.

Programm continuation: Clear alarm with the RESET key. Restart part program

61295 [Channel %1:] Block %2: The value of the 'Axis sequence' parameter is illegal

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

Explanation:

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

Remedy: Correct the "Axis sequence" parameter in the screenform

Programm continuation: Clear alarm with the RESET key. Restart part program

61296 [Channel %1:] Block %2: Blank programmed incorrectly

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

Explanation: The blank has been programmed incorrectly.

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

Remedy: Correct the blank.

Programm continuation: Clear alarm with the RESET key. Restart part program

61297 [Channel %1:] Block %2: Reference for incremental retraction plane missing

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

Explanation: The retraction plane can only be specified in increments if the blank is entered.

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

Remedy: Program the retraction plane in absolute terms.

Programm continuation: Clear alarm with the RESET key. Restart part program

61298 [Channel %1:] Block %2: No work offset entered for main spindle

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

Explanation: No work offset has been specified for the main spindle.

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

Remedy: Specify a work offset for the main spindle in the program header or under Settings.

Programm continuation: Clear alarm with the RESET key. Restart part program

61299 [Channel %1:] Block %2: No work offset entered for counterspindle

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

Explanation: No work offset has been entered for the counterspindle.

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

Remedy: Specify a work offset for the counterspindle in the program header or under Settings.

Programm continuation: Clear alarm with the RESET key. Restart part program

61300 [Channel %1:] Block %2: Probe defective

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61301 [Channel %1:] Block %2: Probe not switching

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

Explanation: The measuring distance was completely traversed, but no switching signal was generated at the measuring input.

Remedy: -Check measuring input.
-Check measuring distance.
-Probe defective.

61302 [Channel %1:] Block %2: Probe - collision

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

Explanation: The measuring probe collided with an obstacle when being positioned.

Remedy: - Check spigot diameter (may be too small)
- Check measuring distance (may be too long)

61303 [Channel %1:] Block %2: Safety margin exceeded %4

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

Explanation: The measuring result differs greatly from the specified value.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check setpoint value and parameter _TSA
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check setpoint value and parameter TSA

61304 [Channel %1:] Block %2: Allowance

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

Explanation: The difference between actual and setpoint measurements is greater than the upper tolerance limit (screen parameter TUL):
- TUL is the upper tolerance limit of the measurement difference.
- TUL is always related to the material, irrespective of whether external or internal machining is involved.
- The hole/pocket is thus too small and the spigot is too large.
- This means that further material can be removed.
- The measuring cycle parameter TUL corresponds to "upper deviation", the usual term used in mechanical engineering for fits and tolerances.

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61305 [Channel %1:] Block %2: Dimension too small

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

Explanation: The difference between the actual and setpoint measurement is less than the lower tolerance limit (screen parameter TLL):
- TLL is the lower tolerance limit of the measurement difference.
- TLL is always related to the material, irrespective of whether external or internal machining is involved.
- The hole/pocket is thus too large and the spigot too small.
- This means that too much material has already been removed.
- The measuring cycle parameter TLL corresponds to "lower deviation", the usual term used in mechanical engineering for fits and tolerances.

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61306 [Channel %1:] Block %2: Permissible measuring difference exceeded

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check setpoint value and parameter DIF
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check setpoint value and parameter DIF

61307 [Channel %1:] Block %2: Incorrect measuring variant %4

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- The value of parameter _MVAR is impermissible.

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- The value of parameter S_MVAR is impermissible.

61308 [Channel %1:] Block %2: Check measuring path

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

Explanation:

Remedy: A traversing path for measuring is generated with a size that can be specified. It describes the maximum distance before and after the expected switching position (workpiece edge) and must have a value greater than 0.

In the AUTOMATIC operating mode:

For 840D sl - to SW 2.6 SP1 and for 828D - to SW 4.3 :

- Check parameter_FA

For 840D sl - from SW 2.7 and for 828D - from SW 4.4 :

- Check parameter DFA

In the JOG mode:

For 840D sl - from SW 2.7 and for 828D - from SW 4.4 :

- Check parameters MD51786: $MNS_J_MEA_T_PROBE_MEASURE_DIST, MD51752: $MNS_J_MEA_M_DIST_TOOL_LENGTH and MD51753: $MNS_J_MEA_M_DIST_TOOL_RADIUS

61309 [Channel %1:] Block %2: Check probe type

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

Explanation:

Remedy: Measurement of workpiece:

Check tool type of measuring probe in the tool management.

When measuring workpiece milling, it is preferred to use tool types 710, 712, 713 or 714. But a type 1xy can also be used.

Type 710 is permitted for workpiece measurement in all measuring cycles. Types 712, 713 and 714 are intended for special measuring tasks.

When measuring workpiece turning, it is preferred to use tool type 580. However, a type 1xy can also be used, but only if setting data $SC_TOOL_LENGTH_TYPE=2 is set.

Measurement of tool:

- For measuring tool milling, no permissible tool probe type is entered in SD54633 $SNS_MEA_TP_TYPE[S_PRNUM-1] or SD54648 $SNS_MEA_TPW_TYPE[S_PRNUM-1] and/or

check the permissible working plane G17...G19 for tool type "Disk".

- "Cube" is the only tool probe type permitted on milling/turning machines:

SD54633 $SNS_MEA_TP_TYPE=0 and SD54648 $SNS_MEA_TPW_TYPE=0

61310 [Channel %1:] Block %2: Scale factor is active

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

Explanation: Scale factor = scaling is active.

Remedy: Switch off the active scale factor in the program. Measuring is not possible with an active scale factor.

61311 [Channel %1:] Block %2: No D number active

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

Explanation: No tool offset for the measuring probe (for workpiece measurement) or no tool offset for the active tool (for tool measurement) is selected.

Remedy: Select the tool's tool edge number D.

61312 [Channel %1:] Block %2: Check measuring cycle number

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

Explanation:

Remedy: The measuring cycle called is impermissible for the current technology.

61313 [Channel %1:] Block %2: Check probe number

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

Explanation:

Remedy: Check parameter S_PRNUM
Measure workpiece: Parameter S_PRNUM 1 to 12
Measure tool: Parameter S_PRNUM 1 to 6

61314 [Channel %1:] Block %2: Check selected tool type

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

Explanation:

Remedy: Measure workpiece:
- Specify new S_PRNUM or recalibrate probe
- Check whether the probe (type 7xx or 5xx) is suitable for the measuring task

Measure tool:
Tool type impermissible for calibration (adjustment) of the tool probe.

61315 [Channel %1:] Block %2: Check position of cutting edge

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

Explanation: When measuring the workpiece in turning, cutting edge positions 7 and 8 are allowed for probe type 580.
Cutting edge positions 5 and 6 are also possible in special applications, such as measuring at the counterspindle.

Remedy: When measuring the workpiece in turning, check the cutting edge position of the probe in the tool list.
When measuring the tool in turning with orientable toolholders, the active cutting edge position of the probe is evaluated.
In this case, the active one must be checked.

61316 [Channel %1:] Block %2: Center and radius cannot be determined

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

Explanation: No circle can be calculated from the measured points, as all measured points lie on a straight line.

Remedy: Program change

61317 [Channel %1:] Block %2: Check number of circle calculation points

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

Explanation: Parameterization faulty; requires 3 or 4 points to calculate the center point.

Remedy: Change parameterization of CYCLE116.

61318 [Channel %1:] Block %2: Check weighting factor

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameter (_K)

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameter (FW)

61319 [Channel %1:] Block %2: Check call parameter CYCLE114

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

Explanation:

Remedy: Check call parameter CYCLE114.

61320 [channel %1:] block %2: check tool name / number

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameters _TNUM, _TNAME.

Bei 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameter S_TNAME .

With active tool management, parameter S_TNAME is empty or
the specified tool name is unknown to the tool management.

61321 [Channel %1:] Block %2: Check WO memory number

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

Explanation:

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

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameter _KNUM

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check the default number of the work offset to be corrected in the measuring cycle screen.
- Check also the parameters S_KNUM and S_KNUM1 in the measuring cycle call interface.

Programm continuation: Clear alarm with the RESET key. Restart part program

61322 [Channel %1:] Block %2: Check 4th digit of _KNUM

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

Explanation: The stated digit of _KNUM includes invalid values. Also check _MVAR.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
Check parameter for tool offset target (_KNUM) and/or measurement variant (_MVAR)

61323 [Channel %1:] Block %2: Check 5th digit of _KNUM

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

Explanation: The stated digit of _KNUM includes invalid values. Also check _MVAR.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
Check parameter for tool offset target (_KNUM) and/or measurement variant (_MVAR)

61324 [Channel %1:] Block %2: Check 6th digit of _KNUM

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

Explanation: The stated digit of _KNUM includes invalid values. Also check _MVAR.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
Check parameter for tool offset target (_KNUM) and/or measurement variant (_MVAR)

61325 [Channel %1:] Block %2: Check measuring axis/offset axis

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameters for measuring axis _MA
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameters for measuring axis (X, Y, Z)

61326 [Channel %1:] Block %2: Check measuring direction

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

Explanation:

Remedy: For 840D sl - to SW 2.6 SP1 and for 828D - to SW 4.3 :
- Milling measuring cycles:
- Parameter for the measuring direction (_MD) has an incorrect value.
For 840D sl - from SW 2.7 and for 828D - from SW 4.4 :
- Milling measuring cycles:
- Check the measuring direction (+ -) entered in the screenform.
For 840D sl and for 828D - from SW 4.6 :
- Milling measuring cycles:
- Check the measuring direction (+ -) entered in the screenform.
- Turning measuring cycles:
- Check the actual pre-position of the workpiece probe with reference to the entered inner or outer measurement.

61327 [Channel %1:] Block %2: Program reset required

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

Explanation: NC reset required.

Remedy: Execute NC reset.

61328 [Channel %1:] Block %2: Check D number

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

Explanation: The D number in parameter _KNUM is larger than the maximum value (MD1805).

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameter for tool offset target (_KNUM)
  Number of the tool edge is greater than that defined in MD1805

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameter for tool offset target (S_KNUM1)
  Number of the tool edge is greater than that defined in MD1805

61329 [Channel %1:] Block %2: Check rotary axis

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

Explanation:

Remedy: No name assigned to the axis number specified in the parameter of the rotary axis, or this axis is not configured as a rotary axis.
For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check MD 20080 and/or MD 30300.

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check MD 20080, MD 30300 and/or MCS 52207 - bit6.

61330 [Channel %1:] Block %2: Coordinate rotation active

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

Explanation: No measuring possible in the rotated coordinate system.

Remedy: Check the conditions for measuring.

61331 [Channel %1:] Block %2: Angle too large, change measuring axis

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Parameter starting angle (_STA) is too large for the measuring axis specified.

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Parameter starting angle (alpha 0) is too large for the measuring axis specified.

Select another measuring axis.

61332 [Channel %1:] Block %2: Modify tool tip position

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

Explanation: The tool tip is below the measuring probe surface (e.g. for a ring gauge or cube).

Remedy: Place the tool above the measuring probe surface.

61333 [Channel %1:] Block %2: Check calibration block number

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

Explanation:

Remedy: Parameter _CALNUM is too large, reduce it to a permissible value
For 840D sl - up to SW 1.x:
- Increase the maximum value of _CVAL[2] in GUD6

For 840D sl/828D - as from SW 2.5:
- Check following machine data: 51601 $MNS_MEA_CAL_EDGE_NUM

61334 [Channel %1:] Block %2: Check safety area

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

Explanation:

Remedy: Check the parameters for the
For 840Dsl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
-_SZA or _SZO
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- XS, YS or ZS

61335 [Channel %1:] Block %2: Reserved

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

Explanation: Alarm triggered: Reserved

Remedy: reserved

61336 [Channel %1:] Block %2: Geometry axes do not exist

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

Explanation: No geometry axes configured.

Remedy: Machine data in MD 20060 must be changed.

61337 [Channel %1:] Block %2: Check measuring input

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

Explanation:

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

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61338 [Channel %1:] Block %2: Positioning speed equal to zero

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

Explanation:

Remedy: For some measuring versions, for example measuring spigots, in addition to the actual measuring paths, intermediate paths are generated that are traversed with a specified feed.
The values for the feed are specified:
- For 840D sl - up to SW 1.x: in parameters _SPEED[1] and _SPEED[2] in GUD6.
- For 840D sl/828D - as from SW 2.5: in setting data 55631 $SCS_MEA_FEED_PLANE_VALUE and 55632 $SCS_MEA_FEED_FEEDAX_VALUE
- For 840D sl/828D - as from SW 4.4 : in the setting data 55634 $SCS_MEA_FEED_PLANE_VALUE and 55636 $SCS_MEA_FEED_FEEDAX_VALUE

61339 [Channel %1:] Block %2:Correction factor for rapid traverse speed = 0

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

Explanation:

Remedy: For 840D sl - up to SW 1.x: Check parameter _SPEED[0] in GUD6
For 840D sl/828D - as from SW 2.5: Check setting data 55630 $SCS_MEA_FEED_RAPID_IN_PERCENT
For 840D sl/828D - as from SW 4.4: Check setting data 55632 $SCS_MEA_FEED_RAPID_IN_PERCENT

61340 [Channel %1:] Block %2: Incorrect alarm number

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

Explanation:

Remedy: Internal error in measuring cycles.

61341 [Channel %1:] Block %2: Probe not calibrated in active plane or wrong calibration data record

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

Explanation:

Remedy: Calibrate the probe before measuring.
Check the number of the probe calibration data field (calibration data record) in parameter S_PRNUM.
Note planes G17, G18 and G19. Only G18 is permitted for workpiece measurement when turning.
Check: Setting data 54611 $SNS_MEA_WP_FEED[S_PRNUM-1] > 0 after calibration

---

61342 [Channel %1:] Block %2: Upgrade NCK software version

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

Explanation:

Remedy: Upgrade NCU software version.

61343 %[[channel %1:] block %2:%]Tool does not exist: %4

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

Explanation:

Remedy: Check tool name.

61344 [Channel %1:] Block %2: Several tools are active

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

Explanation:

Remedy: Remove tool from another spindle.

61345 [Channel %1:] Block %2: D number of tool offset, too many digits

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

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
Reduce the D number in _KNUM, check software or MD of flat D number.

61346 [Channel %1:] Block %2: Distance between starting point and measuring point is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
: - Parameter _SETV[0] or _SETV[1] is empty or less than 0.
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
: - Parameter X1 or X2 is empty or less than 0.

Programm continuation: Clear alarm with the RESET key. Restart part program

61347 [Channel %1:] Block %2: Angle 1st edge - 2nd edge equals 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Parameter following angle (_INCA) is 0.
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Parameter following angle (alpha 1) is 0.

61348 [Channel %1:] Block %2: Angle rel. to reference edge equals 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61349 [Channel %1:] Block %2: Distance upper probe edge - measuring position = 0 for tool radius measurement

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The distance between the upper and lower edges of the tool probe equals 0; relevant for radius measurement.
For 840D sl - up to SW 1.x: Check parameter _TP[x,9]
For 840D sl/828D - as from SW 2.5: Check setting data 54634 $SNS_MEA_TP_CAL_MEASURE_DEPTH

61350 [Channel %1:] Block %2: Feed, speed not programmed for tool measurement with rotating spindle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Measuring feed and/or spindle speed during tool measurement with turning spindle not entered in GUD variable _MFS.
- Check parameter _MFS[0]

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameters F1 and S1

61351 [Channel %1:] Block %2: Tool length or radius is 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Cutter: - Check length and radius of the active tool in the compensation data memory
Drill: - Check length of the active tool in the compensation data memory
- Radius or tip angle of the active tool must be predefined in the compensation data memory

61352 [Channel %1:] Block %2: Path for logfile not permitted

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The specified path for the log file is incorrect.

Remedy: Check parameter _PROTNAME[0]

61353 [Channel %1:] Block %2: Path for logfile not found

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The specified directory does not exist or the specified path is incorrect.

Remedy: Check parameter _PROTNAME[0]

61354 [Channel %1:] Block %2: Log file not found

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: No name specified for the log file.

Remedy: Check parameter _PROTNAME[0]

61355 [Channel %1:] Block %2: Incorrect file type for logfile

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The file extension for the log file is incorrect.

Remedy: Check parameter _PROTNAME[0]

61356 [Channel %1:] Block %2: Logfile is being used

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The log file is already used by an NC program.

Remedy: Check parameter _PROTNAME[1]

61357 %[[Channel %1:] Block %2: %] Not enough NC memory or too many files, directories in the NC

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: There is not enough NC memory available, or too many files or directories in the NC file system.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Delete files whose names are comprised exclusively of numbers, e.g. "201202100938202_MPF", in the directory /_N_WKS_DIR/_N_TEMP_WPD.
Also check whether other files in this directory can be deleted.
Check, and if required, increase MD18320: $MN_MM_NUM_FILES_IN_FILESYSTEM.

Programm continuation: Clear alarm with the RESET key. Restart part program

61358 [Channel %1:] Block %2: Error during reporting: WRITE command %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Internal error

Remedy: Causes of error code %4:
10 The file size specified in MD11420 $MN_LEN_PROTOCOL_FILE has been reached (file is full).
Machine data MD11420 must be set to a value of at least 20 kB, and this sets the maximum size of the log file.
- Increase the value in MD11420 or
- Select log data "new" in cycle150 (data are overwritten) or
- Delete log file or
- Create new log file (change the name of the log file in cycle 150)

Error code 10 occurs only when logging in the passive file system of the NC.
13 The protection level must be greater than or equal to the Write right of the stated log file.
16 Check the path of the log file (invalid external path).
Other errors: See Programming Guide: WRITE command
Save measurement results:
After eliminating the cause of the error, the associated log can be generated with the function "Report last measurement".

61359 [Channel %1:] Block %2: Error recording in table format: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Internal error

Remedy: Call the hotline!

61360 [Channel %1:] Block %2: Error recording in JOG: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Internal error

Remedy: Call the hotline!

61361 [Channel %1:] Block %2: Error reporting last measurement %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Internal error

Remedy: Call the hotline!

61364 [Channel %1:] Block %2: Check distance between measuring points %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - to SW 2.6 SP1 and for 828D - to SW 4.3 :
-In the automatic mode, check parameter clearance, measuring points (_ID).
- In JOG, selected measuring points are identical, redefine measuring points.
For 840D sl - from SW 2.7 and for 828D - from SW 4.4 :
-In the automatic mode, check parameter clearance measuring points (_ID).
- In JOG, selected measuring points are identical, redefine measuring points.

61365 [Channel %1:] Block %2: Check circular feed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
:- Check parameter _RF
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameter SD55640 $SCS_MEA_FEED_CIRCLE

61366 [Channel %1:] Block %2: Direction of rotation for tool measurement with rotating spindle not specified.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 1.x:
- Check parameter _CM[5] in GUD6, permissible values are 3 (corresponds to M3) and/or 4 (corresponds to M4)
For 840D sl/828D - as from SW 2.5:
- Check setting data 54674 $SNS_MEA_CM_SPIND_ROT_DIR, permissible values are 3 (corresponds to M3) and/or 4 (corresponds to M4)

61367 [channel %1:] block %2: parameters %4 are identical

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up tp SW 4.3:
- Specify different positions for the relevant points of _SETV[0...7]
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Specify different positions for the relevant points of P1(X1,Y1), P2(X2,Y2), P3(X3,Y3) and P4(X4,Y4).

61368 [channel %1:] block %2: straights through parameter %4 do not intersect

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up tp SW 4.3:
- Specify different positions for the relevant points of _SETV[0...7]
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Specify different positions for the relevant points of P1(X1,Y1), P2(X2,Y2), P3(X3,Y3) and P4(X4,Y4).

61369 [Channel %1:] Block %2: Position of corner not clearly definable, check parameter %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Define P1 and P2 and/or P3 and P4 so that the intersection of the straights running through these points lies outside the sections formed by P1 and P2 and/or P3 and P4.

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Define P1(X1,Y1) and P2(X2,Y2) and/or P3(X3,Y3) and P4(X4,Y4 so that the intersection of the straights running through these points lies outside the sections formed by P1(X1,Y1) and P2(X2,Y2) and/or P3(X3,Y3) and P4(X4,Y4).

Programm continuation: Clear alarm with the RESET key. Restart part program

61371 [Channel %1:] Block %2: Product of column width and number of columns is greater than %4 characters per line

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check user programming in cust_meaprot

61372 [Channel %1:] Block %2: selected meas.variant requires SPOS-capable spindle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Change measuring variant or check machine equipment.

61373 [Channel %1:] Block %2: No SPOS-capable spindle available

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: - Check configuration/parameterization of the spindle axis
- If the intention to use a 3D workpiece probe at a "non-SPOS capable spindle",
then check the setting of MD 52207 $MCS_AXIS_USAGE_ATTRIB[n], bit 9 (also see Commissioning instructions, cycles)
- If a 3D workpiece probe is attached to the machine outside the spindle,
then check the setting of MD 51740 $MNS_MEA_FUNCTION_MASK, bit 4. (also see Commissioning instructions, cycles)

61374 [Channel %1:] Block %2: Probe not calibrated in axis direction %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Calibrate the probe in the specified axis direction.

61375 [Channel %1:] Block %2: Trigger values of measuring sensor are incompatible

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Trigger values of the tool measuring sensor must be fully described either via the _TP[] / _TPW[] fields or via the setting data
(SD: 54625-54632 or SD: 54640-54647). A mixture of the two variants is not permitted.

61376 [Channel %1:] Block %2: Number of teeth missing from tool parameters

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Enter number of teeth of the tool in the tool management

61377 [Channel %1:] Block %2: Dimensional tolerance %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

61378 [Channel %1:] Block %2: Tool spindle is not the master spindle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Before calling the measuring cycles, the tool spindle must be defined as master spindle (SETMS...).

61379 [Channel %1:] Block %2: Too many teeth

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: As a result of the cycle, tools with a maximum of 100 teeth can be measured.

61380 [Channel %1:] Block %2: Tool probe width too small

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the following machine or setting data: 51781 $MNS_MEA_T_PROBE_THICKNESS[n]

61381 [Channel %1:] Block %2: Coupling the spindle position with coordinate rotation around %4 not executable

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The alarm relates to the cross-measuring tasks measuring cycle function "Coupling of spindle position with coordinate rotation around the infeed axis".
This function requires a positionable spindle.

Remedy: If a positionable spindle is not available for holding the workpiece probe,
the function "Coupling the spindle position with coordinate rotation around the infeed axis"
can be deactivated with SD55740 $SCS_MEA_FUNCTION_MASK, bit 1=0.

61382 [Channel %1:] Block %2 probe is attached to the machine, measuring variant is not possible

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The measuring variant cannot be executed with a probe permanently mounted on the machine.
The selected measuring variant requires rotational positioning/alignment of the probe.

Remedy: Select a measuring variant that does not require rotational positioning/alignment of the probe.

61383 [Channel %1:] Block %2: Rotary axis 1: Diameter tolerance of calibration ball of measurement %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: CYCLE996: See measured diameter in the parameters _OVR[72] to _OVR[74] or the value in the SD $SCS_MEA_KIN_DM_TOL
CYCLE9960: See measured diameter in the parameters _OVR[72] or the value in the SD $SCS_MEA_KIN_DM_TOL

Remedy: Check the calibration data or recalibrate the 3D probe
Check the mechanical structure of the calibration ball in the machine

61384 [Channel %1:] Block %2: Rotary axis 2: Diameter tolerance of calibration ball of measurement %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: CYCLE996: See measured diameter in the parameters _OVR[75] to _OVR[77] or the value in the SD $SCS_MEA_KIN_DM_TOL
CYCLE9960: See measured diameter in the parameters _OVR[75] or the value in the SD $SCS_MEA_KIN_DM_TOL

Remedy: Check the calibration data or recalibrate the 3D probe
Check the mechanical structure of the calibration ball in the machine

61385 [Channel %1:] Block %2: Probe not calibrated in the coordinate system

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Probe has not been calibrated in the coordinate system set in MD52000 $MCS_DISP_COORDINATE_SYSTEM.

Remedy: Calibrate the probe before measurement.

61386 [Channel %1:] Block %2: Invalid protection level for executing measuring cycles %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The protection level is inadequate for the execution of the measuring cycles.

Remedy: See MD51742 $MNS_MEA_ACCESS_EXEC and MD11160 $NM_ACCESS_EXEC_CST

61387 [Channel %1:] Block %2: Internal execution error

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Call the SIEMENS hotline

61401 [Channel %1:] Block %2: Probe does not switch, path reduced by software end position %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The traversing path would extend beyond the software end position on the current measuring axis.
The traversing path has currently been shortened, and does not reach the position specified as a setpoint.
Alarm 61401 is output after 5 unsuccessful contact attempts.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check setpoint value specification.
- Reduce the measuring distance (DFA).
- Place the workpiece further away from the software end position.

Programm continuation: Clear alarm with the RESET key. Restart part program

61402 [Channel %1:] Block %2: Probe collision, path reduced by software end position %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: This alarm is only output with external measurements (web, spigot, ball) in conjunction with a possible collision in the infeed axis.
The collision is caused by the reduction of the path in the measuring axis of the working plane, before lowering the infeed axis to the measuring height.
The probe was switched during the following infeed in the infeed axis.
If the path had not been reduced, the traversing movement would have extended beyond the software end position of the working plane, and the corresponding NCK alarm 10722 would have been triggered.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check setpoint value specification.
- Place the workpiece further away from the software limit position.

Programm continuation: Clear alarm with the RESET key. Restart part program

61403 [Channel %1:] Block %2: Work offset correction not executed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The measurement result contains more than one rotation component around the geometry axes of the coordinate system.
However, only one of the rotation components of a rotary axis can be corrected.
To prevent any consequential position errors, the offset of the selected rotary axis is not performed.
"Correction into the coordinate system" must be selected as the bypass.

61404 [Channel %1:] Block %2: Tool correction not executed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the dependent tool specifications.

61405 [Channel %1:] Block %2: Tool environment does not exist

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Correct the name of the tool environment (_TENV) or create this environment.

61406 [Channel %1:] Block %2: Check DL number

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameter _DLNUM

For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4:
- Check parameter DL

Check the number of the sum offset and that of the setup offset.

61407 [Channel %1:] Block %2: Check 7th digit and higher of _KNUM

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check parameter _KNUM.
- Check the the number of the sum offset and that of the setup offset.

61408 [Channel %1:] Block %2: total offsets not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Set MD 18080, Bit 8=1

61409 [Channel %1:] Block %2: set up offsets not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Set MD 18112, Bit 4=1

61410 [Channel %1:] Block %2 access to nonexistent tool element or property

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The variable to be corrected requires an option or an increase in MD values.

61411 [Channel %1:] Block %2: Check the distribution of measuring points on the plane.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the setpoint and actual values

61412 [Channel %1:] Block %2: channel basic frame not present

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Set MD 28081>0, $P_CHBFRMASK>0

61413 [Channel %1:] Block %2: check setpoint of ball diameter, %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check setpoint value of spherical diameter.

61414 [Channel %1:] Block %2 : distortion of triangle over limit

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the setpoint and actual values

61415 [Channel %1:] Block %2: Check probe / machining plane

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Enter permissible probe for machining plane:
- For 840D sl - up to SW 1.x: Check the variables _TP[x,8] and _TPW[x,8] in GUD6
- For 840D sl/828D - as from SW 2.5: Check the setting data 54633 $SNS_MEA_TP_TYPE and/or 54648 $SNS_MEA_TPW_TYPE

or change the machining plane.

61416 [Channel %1:] Block %2: Probe number is greater than the maximum number of fields

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check parameter S_PRNUM
Measure workpiece: Parameter S_PRNUM 1 to 12
Measure tool: Parameter S_PRNUM 1 to 6

61417 [Channel %1:] Block %2: Probe will collide with the carrier of the reference groove.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Take up collision-free initial position of the axes involved in the measuring process.

61418 [Channel %1:] Block %2: Protocol file too small, check MD11420: LEN_PROTOCOL_FILE.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check MD11420: LEN_PROTOCOL_FILE.
A value of at least 20 kB must be set to use the function "Log measurement results".
This machine data specifies the maximum size of the log file.

61419 [Channel %1:] Block %2: Check probe calibration with reference to center of ball/circumference of ball.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Setting in MD 51740 $MNS_MEA_FUNCTION_MASK bit1 does not match calibration status in
setting data 54610 $SNS_MEA_WP_STATUS_GEN[S_PRNUM] - THOUSANDS digit: 1=circumference 0=TCP (Tool Center Point)

Remedy: Check machine data 51740 $MNS_MEA_FUNCTION_MASK bit1
Remedy:
1. Recalibrate probe (adjust)
2. Change probe number S_PRNUM
3. Adjust MD51740 bit1

61420 [Channel %1:] Block %2: Check calibration of multi/mono probes.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The workpiece probe must be calibrated according to its type and use.

61421 [Channel %1:] Block %2: Software release of measuring cycles or NCK inadequate or set up incorrectly - error code %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: Error code see %4
A -> _OVR[] - parameter field too small. Check GUD definition.
DEF CHAN REAL _OVR[72] (up to MZ06.03.xx.xx =32)
B -> $SCS_MEA_KIN_MODE? SD55645 not set up

61422 [channel %1:] block %2: parameter measuring variant incorrect - error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
A -> CYCLE996 _MVAR = 9x identifiers CYCLE996 measure kinematics or
A -> CYCLE996 _MVAR ONES digit outside the value range 0..4
B -> CYCLE996 _MVAR (parameter for normalization) HUNDRED THOUSANDS digit (dec6) or
B -> CYCLE996 ONE MILLION digit(dec7) outside the value range 0..3
C -> CYCLE996 Measurement version "compute kinematics" active, but rotary axes 1 and/or 2 not measured (see also parameter _OVR[40])
or rotary axis 2 exists and no rotary axis vectors (V2xyz) are set up.
D -> CYCLE996 _MVAR TEN THOUSANDS digit (parameter rotary axis 1,2 or vector chain open, closed) outside the value range 0..3
1 -> CYCLE9960 S_MVAR ONES to TEN THOUSANDS digit outside the value range
2 -> CYCLE9960 S_MVAR TENS digit not compatible with the ONES DIGIT
3 -> CYCLE9960 S_MVAR TENS digit outside the value range 0,2

61423 [Channel %1:] Block %2: Parameter %4 not agreed or not created

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3 :
1. Parameter CYCLE996 _TNUM incorrect or equals zero
2. No swivel data record created -> MD18088 = 0
For 840D sl - as from SW 2.7 and for 828D - as from SW 4.4 :
1. Parameter CYCLE996 S_TC incorrect or equals zero
2. No swivel data record created -> MD18088 = 0
For 840D sl - as from SW 4.7 SP1:
3 $NT_NAME? -> CYCLE996 System variable $NT_NAME not existing
4 TRAFO NO=0? -> CYCLE996 Trafo number = 0 ?
5 TRAFO TYP? -> CYCLE996 Transformation type unequal to 24,40,56
6 S_MVAR? -> CYCLE996 Trafo type OK, but incorrect call from CYCLE9960
7 TRAFO Roundaxis 3? -> CYCLE996 Transformation with 3rd rotary axis for kinematic measuring with CYCLE9960 not allowed
1 (S_TNAME) -> CYCLE9960 Name of swivel data set or transformation not parameterized
2 (KC) -> CYCLE9960 No transformation on the basis of the kinematic chain set up
3 (TC) -> CYCLE9960 No swivel data set found
4 -> CYCLE9960 Compile Cycle E996 not set up

61424 [Channel %1:] Block %2: Parameter %4 for diameter of calibration ball incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check whether the correct diameter of the calibration ball is entered in parameter S_SETV,
or the correct probe ball radius is entered in the tool data of the current workpiece probe.

The mechanical deviation of the position of the current workpiece probe must be minimized by presetting.

61425 [Channel %1:] Block %2: Parameter for measuring axis rotary axis 1 or 2 incorrect - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
A -> Rotary axis number incorrect (1 or 2)
B -> No name agreed for rotary axis 1
C -> Rotary axis vector V1xyz equals zero
D -> No name agreed for rotary axis 2
E -> Rotary axis vector V2xyz equals zero
For 840D sl - as from SW 4.7 SP1:
F VECTOR=0? -> CYCLE996 Orientation vector of the 1st rotary axis = 0
G VECTOR=0? -> CYCLE996 Orientation vector of the 2nd rotary axis = 0
1 -> CYCLE9960 Incorrect number of measurements
2 -> CYCLE9960 Max. number of measurements incorrect per axis 12
3 -> CYCLE9960 Min. number of measurements incorrect per axis 3
4(RA1) -> CYCLE9960 Measuring range rotary axis 1 too small
4(RA2) -> CYCLE9960 Measuring range rotary axis 2 too small
5(RA1) -> CYCLE9960 Measuring range rotary axis1 too large
5(RA2) -> CYCLE9960 Measuring range rotary axis 2 too large
6 -> CYCLE9960 Number of measurements incorrect, min. per axis 2 for interpolation points
7 -> CYCLE9960 Mod_Range Measuring range covers the modulo range, and therefore must begin at the starting position
The permissible measuring range of a rotary axis is from 10 to 360 degrees

61426 [Channel %1:] Block %2: Sum of the active offsets does not equal zero - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
Check the overview of active offsets ($P_ACTFRAME)
Delete offsets in the rotary axes
A -> sum of the translatory offsets of geometry axes <> 0
B -> sum of the fine offsets of the geometry axes <> 0
C -> sum of the rotary components of the geometry axes <> 0
D -> sum of the translatory offsets of rotary axis 1 <> 0
E -> sum of the translatory offsets of rotary axis 2 <> 0

61427 [Channel %1:] Block %2: Tool data of the active workpiece probe incorrect or inactive - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
A -> Workpiece probe (or tool edge) inactive
B -> Length L1 of the workpiece probe = 0

61428 [Channel %1:] Block %2: Error while creating log file - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
A -> Number of log files in the current directory > 99
B -> Log files too long. Rename or delete log files,
check MD11420 $MN_LEN_PROTOCOL_FILE!

61429 [Channel %1:] Block %2: Measuring axis (rotary axis 1 or 2) not in basic or intended position - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Error causes: See Error code %4
A -> Rotary axis 1 not in basic position for 1st measurement
B -> Rotary axis 2 not in basic position for 1st measurement
C -> Rotary axis 2 not rotated for the 2nd or 3rd measurement with reference to the 1st measurement -> see parameter _OVR[63 to 65]
D -> Rotary axis 1 not rotated for the 2nd or 3rd measurement with reference to the 1st measurement -> see parameter _OVR[60 to 62]
1 (RA1) -> CYCLE9960 Rotary axis 1 initially not in basic position
1 -> CYCLE9960 Rotary axis 1 or 2 initially not in basic position

61430 [Channel %1:] Block %2: Kinematic vectors not computed - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Causes of error: See error code %4
A -> Plausibility of the input points PM1, PM2, PM3 not fulfilled, resulting side lengths must not be equal to zero
(Notice: even in the case of side lengths not equal to zero, there is a risk of not being able to form a triangle => check PM1...3!)
B -> Enclosed angle at PM1 between the spread vectors PM1PM2 and PM1PM3 is equal to 0.
Starting points do not form a triangle.
C -> Enclosed angle at PM2 between the spread vectors PM2PM1 and PM2PM3 is equal to 0.
Starting points do not form a triangle.
D -> Enclosed angle at PM3 between the spread vectors PM3PM1 and PM3PM2 is equal to 0.
Starting points do not form a triangle.
E -> Normalizing interpolation point: Invalid axis name defined for computation
F -> Normalizing interpolation point: Invalid plane defined for computation
G -> If the computed angle is smaller than the limiting value of the angular segment of the rotary axis in parameter TVL. The computed limit angle and the name of the rotary axis are displayed.
With values of TVL < 20 degrees, inaccuracies are to be expected as a result of measuring inaccuracies in the micrometer range of the probe.
Example: "61429 .. G Axis:C->TVLmin=12.345"
Remedy: Adjust angular value of rotary axis in user program or parameter TVL.
Note regarding 840 Dsl sb SW 4.7. SP1
With complete measurement of kinematics with CYCLE9960, the value for the limit angles is entered in SD $SCS_MEA_KIN_MIN_ANG_TRIANGLE (minimum internal angle of the measuring triangle)
"CC Option ? " -> Option compile cycle "Measurement of kinematics" not set
"$MN_CC_ACTIVE_IN_CHAN_C996[0] ? " -> machine data for compile cycle not set
"License ? " -> License for "Measurement of kinematics" not set
For 840D sl - as from SW 4.7 SP1:
When measuring head kinematics without offset (spindle extension), an alternative calculation is performed with error message A or G. This applies on condition that the rotary axis to be measured has been repositioned
H -> Conditions for alternative calculation not fulfilled (rotary axis not positioned -> see _OVR[60] to _OVR]62] or SD55648 $SCS_MEA_KIN_MIN_ANG_POS smaller than the difference of the rotary axis positions in _OVR[60] to _OVR]62]
I -> Conditions for alternative calculation not fulfilled (rotary axis not positioned -> see _OVR[60] to _OVR]62]

61431 [Channel %1:] Block %2: Kinematic vectors not corrected - Error code: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Error causes: see error code %4
A -> Correction not possible with active transformation

61440 [Channel %1:] Block %2: Position of cutting edge cannot be determined

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A turning tool with a cutting edge position between 1 and 8 must be used as the tool type.
Check the entered cutting edge position with reference to the basic position of the tool carrier.

61441 [Channel %1:] Block %2: Position of cutting edge is not in the machining plane.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The position of the cutting edge of the turning tool (cutting tip) is no longer in the machining plane (interpolation plane), this can be caused, for example, by a tool carrier with orientation capability. Correct the tool carrier position.

61442 [Channel %1:] Block %2: Tool carrier not parallel to the geometry axes

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: After positioning the orientable tool carrier, tool lengths L1, L2 and L3 are not parallel to the geometry axes.
Check the positioning behavior of the rotary axes (blocking) of the tool carrier.

61443 [Channel %1:] Block %2: Advance angle %4 or greater/less than +/-90° or +/-120°

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: For 840D sl - up to SW 2.6 SP1 and for 828D - up to SW 4.3:
- Check the value in the parameter advance angle _INCA!
- If 3-point measurement is selected, _INCA must not be greater/less than +/-120°, and with 4-point measurement _INCA must not be greater/less than +/-90°.
- The advance angle _INCA must always be parameterized unequal to "zero".

For 840D sl - aa from SW 2.7 and for 828D - as from SW 4.4:
- Check the value in the parameter advance angle alpha 1!
- If 3-point measurement is selected, alpha 1 must not be greater/less than +/-120° and with 4-point measurement, alpha 1 must not be greater/less than +/-90°!
- The advance angle alpha 1 must always be parameterized unequal to "zero".

61444 [Channel %1:] Block %2: Current measuring speed is not identical to the calibration speed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: The relevant calibration speed is also stored in each calibration data record.
The current measuring speed with feedrate override of 100% is not equal to the calibration speed.
After calibration, the calibration speed value must be in the following setting data.
For workpiece measurement:
SD 54611 $SNS_MEA_WP_FEED[S_PRNUM-1] > 0
For tool measurement:
SD 54636 $SNS_MEA_TP_FEED[S_PRNUM-1] > 0 for calibration in the machine coordinate system
SD 54651 $SNS_MEA_TPW_FEED[S_PRNUM-1] > 0 for calibration in the workpiece coordinate system
Recalibrate probe (adjust) or specify new S_PRNUM.

61445 [Channel %1:] Block %2: Check holder angle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the entry for the holder angle in the tool offset.
For cutting edge positions 1-4, the holder angle must be greater than or equal to 90° and less than 180°,
for cutting edge positions 5-8, it must be greater than 0° and less than 90°.

61446 [Channel %1:] Block %2: Check insert angle and clearance angle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the entry for the insert angle / clearance angle in the tool offset!

61447 [Channel %1:] Block %2: Reference point, measuring direction, check tool default setting of probe

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: This alarm is only output for measurement in JOG with calibration of probe length,
on a reference piece or on the sphere.
An implausible result has occurred, comparable with overrunning
the safe area in automatic mode.

Remedy: Check the specifications stated in the alarm text.

61501 [Channel %1:] Block %2: Simulation is active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Reset simulation

Programm continuation: Clear alarm with the RESET key. Restart part program

61502 [Channel %1:] Block %2: No tool offset active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A tool number must be programmed

Programm continuation: Clear alarm with the RESET key. Restart part program

61503 [Channel %1:] Block %2: tool nose radius compensation left or right

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A tool offset value has to be programmed

Programm continuation: Clear alarm with the RESET key. Restart part program

61504 [Channel %1:] Block %2: _KNG incorrect for setup

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61505 [Channel %1:] Block %2: retraction path is smaller than 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Increase retraction path

Programm continuation: Clear alarm with the RESET key. Restart part program

61506 [Channel %1:] Block %2: infeed path is smaller than 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Increase infeed path

Programm continuation: Clear alarm with the RESET key. Restart part program

61507 [Channel %1:] Block %2: safety clearance is smaller than 1mm

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61508 [Channel %1:] Block %2: Incorrect default setting for shoulder position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61509 [Channel %1:] Block %2: Incorrect default setting for dresser position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61510 [Channel %1:] Block %2: Test run feed is active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Switch off test run feed

Programm continuation: Clear alarm with the RESET key. Restart part program

61511 [Channel %1:] Block %2: Incorrect shoulder position or tool edge D1/D2

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61512 [Channel %1:] Block %2: Incorrect longitudinal position

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61513 [Channel %1:] Block %2: Dresser left and inclined grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61514 [Channel %1:] Block %2: Grinding wheel type missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61515 [Channel %1:] Block %2: Retraction path is smaller than or equal to dressing amount

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Change retraction path

Programm continuation: Clear alarm with the RESET key. Restart part program

61517 [Channel %1:] Block %2: Angle of inclined grinding wheel missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Enter angle under $TC_TPG8

Programm continuation: Clear alarm with the RESET key. Restart part program

61518 [Channel %1:] Block %2: Shoulder height of the grinding wheel must be greater than the grinding wheel radius

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Change shoulder height or grinding wheel radius

Programm continuation: Clear alarm with the RESET key. Restart part program

61519 %[[Channel %1:] Block %2: %]Incorrect type of machining

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Assign a value between 1 and 3 to parameter B_ART

Programm continuation: Clear alarm with the RESET key. Restart part program

61520 [Channel %1:] Block %2: Additional offsets not set

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Set MD18094 MM_NUM_CC_TDA_PARAM=10

Programm continuation: Clear alarm with the RESET key. Restart part program

61521 [Channel %1:] Block %2: Current grinding wheel too wide

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Reduce width of grinding wheel

Programm continuation: Clear alarm with the RESET key. Restart part program

61522 [Channel %1:] Block %2: Overlap is greater than or equal to the actual grinding wheel width

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Reduce overlap

Programm continuation: Clear alarm with the RESET key. Restart part program

61523 [Channel %1:] Block %2: Zero signal of calipers missing

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check calipers signal

Programm continuation: Clear alarm with the RESET key. Restart part program

61524 [Channel %1:] Block %2: Incorrect oblique angle

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Oblique plunge angles must be >-90° and <90°

Programm continuation: Clear alarm with the RESET key. Restart part program

61525 [Channel %1:] Block %2: Incorrect grinding wheel type

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Change grinding wheel type $TC_TPC1

Programm continuation: Clear alarm with the RESET key. Restart part program

61526 [Channel %1:] Block %2: Workpiece radius = 0

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Enter workpiece radius > 0

Programm continuation: Clear alarm with the RESET key. Restart part program

61527 [Channel %1:] Block %2: Grinding wheel radius is greater than or equal to the workpiece radius

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Change grinding wheel radius or workpiece radius

Programm continuation: Clear alarm with the RESET key. Restart part program

61529 [Channel %1:] Block %2: Dimensional notation INCH programmed

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Basic system MD $MN_SCALING_SYSTEM_IS_METRIC does not correspond to programmed G command (G group 13).

Programm continuation: Clear alarm with the RESET key. Restart part program

61530 [Channel %1:] Block %2: Default longitudinal position incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check longitudinal position parameter

Programm continuation: Clear alarm with the RESET key. Restart part program

61531 [Channel %1:] Block %2: Longitudinal position not registered in Z

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Increase infeed path parameter

Programm continuation: Clear alarm with the RESET key. Restart part program

61532 [Channel %1:] Block %2: Value for _LAGE is incorrect

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Correct parameter content for _LAGE

Programm continuation: Clear alarm with the RESET key. Restart part program

61533 [Channel %1:] Block %2: No length L1 entered under D...

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Enter length L1 in the tool offset D of the grinding wheel

Programm continuation: Clear alarm with the RESET key. Restart part program

61540 [Channel %1:] Block %2: Incorrect D number / dresser D field active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A tool D number must be programmed that is < _GC_DNUM

Programm continuation: Clear alarm with the RESET key. Restart part program

61541 [Channel %1:] Block %2: Incorrect grinding wheel type entered

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Select a valid grinding wheel type in tool management

Programm continuation: Clear alarm with the RESET key. Restart part program

61542 [Channel %1:] Block %2: Incorrect grinding wheel reference point selected when selecting the dresser coordinate system

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A tool D number must be programmed that is < _GC_DNUM

Programm continuation: Clear alarm with the RESET key. Restart part program

61543 [Channel %1:] Block %2: Incorrect dresser selected when selecting the dresser coordinate system

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: A dresser number >0 and <4 must be selected

Programm continuation: Clear alarm with the RESET key. Restart part program

61544 [Channel %1:] Block %2: Grinding wheel diameter worn down

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: New grinding wheel required, or check limit values in the grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61545 [Channel %1:] Block %2: Width of grinding wheel worn down

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: New grinding wheel required, or check limit values in the grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61546 [Channel %1:] Block %2: Dresser %4, wear limit length 1 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: New dresser required, or check limit values of dresser

Programm continuation: Clear alarm with the RESET key. Restart part program

61547 [Channel %1:] Block %2: Dresser %4, wear limit length 2 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: New dresser required, or check limit values of dresser

Programm continuation: Clear alarm with the RESET key. Restart part program

61548 [Channel %1:] Block %2: Dresser %4, wear limit length 3 reached

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: New dresser required, or check limit values of dresser

Programm continuation: Clear alarm with the RESET key. Restart part program

61549 [Channel %1:] Block %2: Incorrect dresser type selected

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check dresser type on input

Programm continuation: Clear alarm with the RESET key. Restart part program

61550 [Channel %1:] Block %2: Swiveling of plane not possible with active grinding tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Swiveling of the plane is not possible with an active grinding tool.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Load a milling tool before you swivel the plane.
The alarm can be suppressed using SD 55410 $SCS_MILL_SWIVEL_ALARM_MASK.

Programm continuation: Clear alarm with the RESET key. Restart part program

61551 [Channel %1:] Block %2: Setting of milling tool not possible with active grinding tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Setting of milling tools is not possible with an active grinding tool.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Load a milling tool, before you call positioning.

Programm continuation: Clear alarm with the RESET key. Restart part program

61552 [Channel %1:] Block %2: Alignment of milling tool not possible with active grinding tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Alignment of milling tools is not possible with an active grinding tool.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Load a milling tool before you call alignment.

Programm continuation: Clear alarm with the RESET key. Restart part program

61553 [Channel %1:] Block %2: Alignment of grinding tool only possible with active grinding tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Alignment of grinding tools is only possible with an active grinding tool.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Load a grinding tool before calling alignment.

Programm continuation: Clear alarm with the RESET key. Restart part program

61555 [Channel %1:] Block %2: Diameter of grinding wheel ==0, GWPS cannot be calculated

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check diameter

Programm continuation: Clear alarm with the RESET key. Restart part program

61556 [Channel %1:] Block %2: Impossible chamfer and radius of left edge of wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61557 [Channel %1:] Block %2: Impossible chamfer and radius of right edge of wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61558 [Channel %1:] Block %2: Chamfer / radius + shoulder height are larger than the retraction height of the left edge of the grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61559 [Channel %1:] Block %2: Chamfer / radius + shoulder height are larger than the retraction height of the right edge of the grinding wheel

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61560 [Channel %1:] Block %2: Infeed in Z direction too big per stroke, or wheel too narrow

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Reduce infeed path parameter or use other tool

Programm continuation: Clear alarm with the RESET key. Restart part program

61561 [Channel %1:] Block %2: Feed left wheel edge is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61562 [Channel %1:] Block %2: Feed right wheel edge is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61563 [Channel %1:] Block %2: Feed on the diameter is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61564 %[[Channel %1:] Block %2: %]Feed insertion is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61565 [Channel %1:] Block %2: Feed dressing is smaller than or equal to zero

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61566 [Channel %1:] Block %2: Chamfer / radius is larger than the grinding wheel width

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check values in grinding wheel data

Programm continuation: Clear alarm with the RESET key. Restart part program

61567 [Channel %1:] Block %2: %4 Total infeed depth and sum of the infeed must have the same sign

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Remedy: Check the signs of the infeeds

Programm continuation: Clear alarm with the RESET key. Restart part program

61568 %[[Channel %1:] Block %2: %]Error during form-truing %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: -103: Machining not possible
-121: Cancellation due to lack of memory

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: -103: Check grinding wheel contour or position of dresser does not match contour
-121: Check contour of the grinding wheel and technological data
Call the SIEMENS hotline

Programm continuation: Clear alarm with the RESET key. Restart part program

61569 %[[Channel %1:] Block %2: %]Machining plane different in new profiling and continue profiling

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - The machining plane must be the same for new profiling and continue profiling.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61570 %[[Channel %1:] Block %2: %]Profiling type is different in new profiling and continue profiling

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - The type of profiling (parallel to the axis, parallel to the contour) must be the same for new profiling and continue profiling.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61571 %[[Channel %1:] Block %2: %]Infeed direction is different for new profiling and continue profiling

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - The infeed direction must be the same for new profiling and continue profiling.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61572 %[[Channel %1:] Block %2: %]Grinding wheel is not suitable for continue profiling

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - The same grinding wheel must be used for continue profiling as for new profiling.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61573 %[[Channel %1:] Block %2: %]Edge of grinding wheel is not suitable for continue profiling

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - The same edge of the grinding wheel must be used for continue profiling as for new profiling.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61574 %[[Channel %1:] Block %2: %]Error in contour %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: -1: Contour is not continuous

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of the contour
-1: Contour must be continuous in the machining axis

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61575 %[[Channel %1:] Block %2: %]G0 blocks in the grinding wheel contour description

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: G0 blocks are included in the grinding wheel contour description.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - G0 blocks may not be included in the contour description.
- Check whether approach movements to and retraction movements from the grinding wheel are included in the controur description. If this is the case, remove them.

Programm continuation: Clear alarm with the RESET key. Restart part program

---

61576 %[[Channel %1:] Block %2: %]The grinding wheel profile cannot be exactly machined with the active tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The grinding wheel profile cannot be exactly machined with the tool angles (clearance and pitch angles) of the active dressing tool.
Residual material would be left when using the current tool angles.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check tool angles (clearance and pitch angles) of the active dressing tool resp. contour.
Use another dressing tool.

Programm continuation: Clear alarm with the RESET key. Restart part program

61577 %[[Channel %1:] Block %2: %]No tool carrier %4 is active

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: - No tool carrier is active

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Activate tool carrier

Programm continuation: Clear alarm with the RESET key. Restart part program

61578 [Channel %1:] Block %2: B axis kinematics (grinding technology) either not started up or started up incorrectly in swivel cycle - error code: %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Remedy: Causes of error:
1. Error code = B789 -> B axis in start-up of swivel cycle (kinematics) not activated
(789 corresponds to $TC_CARR7[n], n ... number of the swivel data record)

Programm continuation: Clear alarm with the RESET key. Restart part program

61601 [Channel %1:] Block %2: Finished part diameter too small

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Remedy: Check parameter SPD or DIATH.

Programm continuation: Clear alarm with the RESET key. Restart part program

61602 [Channel %1:] Block %2: Tool width incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Plunge cutter is larger than the programmed groove width.

Remedy: Check tool or change program.

Programm continuation: Clear alarm with the RESET key. Restart part program

61603 [Channel %1:] Block %2: Recess type incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Radii/chamfers at the groove base do not match the groove width. Face groove on a contour element running parallel to the longitudinal axis is not possible.

Remedy: Check parameter VARI.

Programm continuation: Clear alarm with the RESET key. Restart part program

61604 [Channel %1:] Block %2: Active tool violates programmed contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Contour violation in the relief cut elements due to the tool clearance angle of the tool used.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Use a different tool or check the contour subroutine.

Programm continuation: Clear alarm with the RESET key. Restart part program

61605 [Channel %1:] Block %2: Contour incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Illegal relief cut element detected.

Remedy: Check contour program.

Programm continuation: Clear alarm with the RESET key. Restart part program

61606 [Channel %1:] Block %2: Error during contour preparation

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: An error has been found on conditioning the contour. This alarm is always related to one of NCK alarms 10930...10934, 15800 or 15810.

Remedy: Check contour subroutine.

Programm continuation: Clear alarm with the RESET key. Restart part program

61607 [Channel %1:] Block %2: Starting point incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The starting point reached before the cycle call does not lie outside the rectangle described by the contour subroutine.

Remedy: Check starting point prior to cycle call.

Programm continuation: Clear alarm with the RESET key. Restart part program

61608 [Channel %1:] Block %2: Incorrect tool point direction programmed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: A cutting edge position 1...4, matching the undercut form, must be programmed.

Programm continuation: Clear alarm with the RESET key. Restart part program

61609 [Channel %1:] Block %2: Shape incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Remedy: Check parameter for the undercut form or groove form or pocket.

Programm continuation: Clear alarm with the RESET key. Restart part program

61610 [Channel %1:] Block %2: No infeed depth programmed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Remedy: Check parameter MID.

Programm continuation: Clear alarm with the RESET key. Restart part program

61611 [Channel %1:] Block %2: No point of intersection found

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: No intersection could be calculated with the contour.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check contour programming or modify infeed depth.

Programm continuation: Clear alarm with the RESET key. Restart part program

61612 [Channel %1:] Block %2: Synchronized thread machining not possible

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Remedy: Check the preconditions for synchronized thread machining:
- It is not permissible that a toolholder is active.
- It is not permissible that transformation is active.
- It is not permissible that rotation is active.

If required, deselect thread synchronization.

61613 [Channel %1:] Block %2: Undercut position incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check value in parameter _VARI.

Programm continuation: Clear alarm with the RESET key. Restart part program

61614	[Channel %1:] Block %2: %4 Z mirroring in WO for main spindle not permitted
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	WO for main spindle machining must not have Z mirroring.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Deselect Z mirroring in the WO used.
Programm continuation:	Clear alarm with the RESET key. Restart part program

61615	[Channel %1:] Block %2: %4- Mirroring in the work offset is not permitted for the counterspindle
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	The work offset for counterspindle machining must not have any Z mirroring.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Deselect Z mirroring in the WO used.
Programm continuation:	Clear alarm with the RESET key. Restart part program

61616	[Channel %1:] Block %2: Current cutting edge position %4 impermissible
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Cutting edge positions 1 to 4 are permissible for corner stock removal.
Programm continuation:	Clear alarm with the RESET key. Restart part program

61617	[Channel %1:] Block %2: Maximum spindle speed for the main spindle not entered
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	The maximum spindle speed for the main spindle was not entered.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Enter the maximum spindle speed for the main spindle.
Programm continuation:	Clear alarm with the RESET key. Restart part program

61618 [Channel %1:] Block %2: Main spindle has not been set up

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: A main spindle has not been set up.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Set up main spindle in MD52206 $MCS_AXIS_USAGE.

Programm continuation: Clear alarm with the RESET key. Restart part program

61619 [Channel %1:] Block %2: Main spindle has not been correctly set up

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The main spindle has not been correctly set up.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check the commissioning of the main spindle.
Check machine data MD20070 $MC_AXCONF_MACHAX_USED, MD20080 $MC_AXCONF_CHANAX_NAME_TAB and MD52207 $MCS_AXIS_USAGE_ATTRIB bit 8.

Programm continuation: Clear alarm with the RESET key. Restart part program

61620 [Channel %1:] Block %2: %4-Mirroring for the linear axis of the counter spindle not permitted

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: It is not permissible that the linear axis of the counterspindle machining has Z mirroring.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Deselect Z mirroring in the WO used.

Programm continuation: Clear alarm with the RESET key. Restart part program

61621 [Channel %1:] Block %2: Angle of opening of the convex thread too high

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Convexity of the thread is too high.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter XS or RS

Programm continuation: Clear alarm with the RESET key. Restart part program

61622	[Channel %1:] Block %2: Toolcarrier for turning not set up
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	A toolcarrier was not set up for turning.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Set up a tool with carrier with the rotary axes: B axis and tool spindle. Set the identifier "B axis kinematics".
Programm continuation:	Clear alarm with the RESET key. Restart part program

61623	[Channel %1:] Block %2: A toolcarrier has not been set up for milling on the main spindle
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	A toolcarrier has not been setup for a milling on the main spindle.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Set up a toolcarrier with rotary axes: B axis and main spindle. Do not set the identifier "B axis kinematics".
Programm continuation:	Clear alarm with the RESET key. Restart part program

61624	[Channel %1:] Block %2: A toolcarrier has not been set up for milling on the counterspindle
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	A toolcarrier has not been set up for milling on the counterspindle.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Set up a toolcarrier with the rotary axes: B axis and counterspindle. Do not set the identifier "B axis kinematics".
Programm continuation:	Clear alarm with the RESET key. Restart part program

61625	[Channel %1:] Block %2: Programmed angular value is not in the grid of the Hirth gearing: %4
Parameters:	%1 = Channel number %2 = Block number, label
Explanation:	An angular value that is not in the grid of the Hirth gearing was programmed.
Reaction:	Interpreter stop NC Start disable in this channel. Interface signals are set. Alarm display.
Remedy:	Program an angular value that is in the grid of the Hirth gearing.
Programm continuation:	Clear alarm with the RESET key. Restart part program

61626 [Channel %1:] Block %2: With the programmed angular values, the tool tip is not located in the turning plane %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Angular values were programmed where the tool tip is not located in the turning plane.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Program suitable angular values or use the programming based on "Beta" and "Gamma".

Programm continuation: Clear alarm with the RESET key. Restart part program

61627 [Channel %1:] Block %2: Rotation of the turning tool not possible as the tool holder for turning tools is fixed.

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Rotation of the turning tool not possible as the tool holder for turning tools is fixed.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Select Use "fixed"

Programm continuation: Clear alarm with the RESET key. Restart part program

61695 %[[Channel %1:] Block %2: %]Parameter R123 incorrectly programmed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter R123

Programm continuation: Clear alarm with the RESET key. Restart part program

61696 %[[Channel %1:] Block %2: %]Parameter R122 too high

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter R122

Programm continuation: Clear alarm with the RESET key. Restart part program

61697 %[[Channel %1:] Block %2: %]Parameter R122 too low

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter R122

Programm continuation: Clear alarm with the RESET key. Restart part program

61698 %[[Channel %1:] Block %2: %]Parameter R122 has incorrect sign

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter R122

Programm continuation: Clear alarm with the RESET key. Restart part program

61699 %[[Channel %1:] Block %2: %]Parameter R121 has incorrect sign

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter R121

Programm continuation: Clear alarm with the RESET key. Restart part program

61700 %[[Channel %1:] Block %2: %]Name of program to be generated is missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter PRG

Programm continuation: Clear alarm with the RESET key. Restart part program

61701 %[[Channel %1:] Block %2: %]Contour %4 does not exist

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter CON
- Check contour call
- Check whether the contours exist in the program storage (workpieces, subroutines or part programs)

Programm continuation: Clear alarm with the RESET key. Restart part program

61702 %[[Channel %1:] Block %2: %]Label %4 not existing in machined part contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check whether the labels exist in the machined part contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61703 %[[Channel %1:] Block %2: %]Label %4 not existing in the blank contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check whether the labels exist in the blank contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61704 %[[Channel %1:] Block %2: %]Machined part contour missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contour call (CYCLE62)

Programm continuation: Clear alarm with the RESET key. Restart part program

61705 %[[Channel %1:] Block %2: %]Blank contour missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contour call

Programm continuation: Clear alarm with the RESET key. Restart part program

61706 %[[Channel %1:] Block %2: %]Error in machined part contour %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of machined part contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61707 %[[Channel %1:] Block %2: %]Error in the blank contour %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of the blank contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61708 %[[Channel %1:] Block %2: %]Too many contours specified

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check number of contours
- Max. two contours (machined part and blank contours)
- Min. one contour (machined part contour)

Programm continuation: Clear alarm with the RESET key. Restart part program

61709 %[[Channel %1:] Block %2: %]Cutting edge radius too small

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check cutting edge radius of the tool in tool management

Programm continuation: Clear alarm with the RESET key. Restart part program

61710 %[[Channel %1:] Block %2: %]Calculation has been canceled

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Calculation has been canceled by PI service; please try again

Programm continuation: Clear alarm with the RESET key. Restart part program

61711 %[[Channel %1:] Block %2: %]Infeed D is larger than the tip width of the tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check infeed D in connection with the tip width of the tool in tool management.

Programm continuation: Clear alarm with the RESET key. Restart part program

61712 %[[Channel %1:] Block %2: %]Infeed DX or DZ is larger than tip length of tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check infeed DX or DZ in connection with tip length of the tool in tool management.

Programm continuation: Clear alarm with the RESET key. Restart part program

61713 %[[Channel %1:] Block %2: %]Tool radius larger than half the tip width

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check tool radius and tip width of tool (plunge cutter, cutting tool)

Programm continuation: Clear alarm with the RESET key. Restart part program

61714 %[[Channel %1:] Block %2: %]Error during contour turning %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: -103: Machining not possible
-121: Cancellation due to lack of memory

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: -103: Check contour or the position of the tool does not match the contour
-121: Check contour and technological data
Call the SIEMENS hotline

Programm continuation: Clear alarm with the RESET key. Restart part program

61730 %[[Channel %1:] Block %2: %]Machining range outside delimitation

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check machining range and delimitation

Programm continuation: Clear alarm with the RESET key. Restart part program

61731 %[[Channel %1:] Block %2: %]Unable to determine contour direction

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contours
- Check whether the contour starting point exists

Programm continuation: Clear alarm with the RESET key. Restart part program

61732 %[[Channel %1:] Block %2: %]No material available for machining

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of the blank and machined part contour, particularly their position towards each other

Programm continuation: Clear alarm with the RESET key. Restart part program

61733 %[[channel %1:] block %2: %]Cutting edge not compatible with machining direction

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programmed machining direction in connection with cutting edge position of the tool

Programm continuation: Clear alarm with the RESET key. Restart part program

61734 %[[Channel %1:] Block %2: %]Machined part contour is outside the blank contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of the machined part and blank contour, particularly their position towards each other

Programm continuation: Clear alarm with the RESET key. Restart part program

61735 %[[Channel %1:] Block %2: %]Infeed D larger than the tip length of the tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check infeed D in relation to the tip length of the tool in tool management

Programm continuation: Clear alarm with the RESET key. Restart part program

61736 %[[Channel %1:] Block %2: %]Cutting depth greater than maximum depth of cut of the tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61737 %[[Channel %1:] Block %2: %]Cutting depth smaller than minimum depth of cut of the tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61738 %[[Channel %1:] Block %2: %]Incorrect cutting edge position

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check cutting edge position in tool management

Programm continuation: Clear alarm with the RESET key. Restart part program

61739 %[[Channel %1:] Block %2: %]Blank must be closed contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check whether the blank contour is closed

Programm continuation: Clear alarm with the RESET key. Restart part program

61740 %[[Channel %1:] Block %2: %]Collision through approach

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Select the starting position to enable collision-free approach of the contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61741 %[[Channel %1:] Block %2: %]Axis in negative range

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check the position of the axis in the ordinate

Programm continuation: Clear alarm with the RESET key. Restart part program

61742 %[[Channel %1:] Block %2: %]Retraction plane %4 is within the machining range

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - On internal machining check machining range in relation to the entered retraction distance ($SCS_TURN_ROUGH_I_RELEASE_DIST)

Programm continuation: Clear alarm with the RESET key. Restart part program

61743 %[[channel %1:] block %2: %]Guide channel for two-channel cutting missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if a guide channel has been defined

Programm continuation: Clear alarm with the RESET key. Restart part program

61744 %[[channel %1:] block %2: %]Following channel for two-channel cutting missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if a following channel has been defined

Programm continuation: Clear alarm with the RESET key. Restart part program

61745 %[[channel %1:] block %2: %]Two-channel cutting in two guide channels (%4) active

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if two guide channels are active
- Check if two-channel cutting is active simultaneously in more than two channels
- Two channels only may be active: a guide and a following channel

Programm continuation: Clear alarm with the RESET key. Restart part program

61746 %[[channel %1:] block %2: %]Two-channel cutting active already in the channels (%4)

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if two-channel cutting is active simultaneously in more than two channels.
- Two channels only may be active simultaneously: a guide and a following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61747 %[[channel %1:] block %2: %]Incorrect guide channel for two-channel cutting (%4)

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if the program of the guide channel runs in the channel selected via the program of the following channel in parameter partner channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61748 %[[channel %1:] block %2: %]Machining plane different in guide and following channel

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Machining plane must be the same in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61749 %[[channel %1:] block %2: %]Technology different in guide and following channel

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Technology (cutting / residual, plunge-cutting / residual, plunge-turning / residual) must be the same in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61750 %[[channel %1:] block %2: %]Machining different in guide and following channel

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Machining (rough cutting / finish cutting) must not be different in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61751 %[[Channel %1:] Block %2: %]The dimension system is different in the leading and following channels

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Dimension system (G group: 13 (G70, G71, G700, G710)) must be the same in both the leading and following channels.

Programm continuation: Clear alarm with the RESET key. Restart part program

61752 %[[Channel %1:] Block %2: %]Cutting edge positions or cutting directions of the tools are different

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Cutting edge position and cutting direction of the tools must be the same in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61753 %[[channel %1:] block %2: %]Difference in tool radii too large

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Tool radii may not be greater than the finishing allowance at maximum for rough cutting.

Programm continuation: Clear alarm with the RESET key. Restart part program

61754 %[[channel %1:] block %2: %]Tool radii must have same size for rough cutting

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if tool radii have the same size in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61755 %[[channel %1:] block %2: %]Plate widths are different

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check if plate widths of recessing tools are the same size in guide and following channel.

Programm continuation: Clear alarm with the RESET key. Restart part program

61756 **%[[Channel %1:] Block %2: %]Multichannel machining not possible because of collision of the tools.**

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Multichannel machining with the programmed parameters would lead to a collision of the tools.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Adjust parameter (DCH offset) so that mutichannel machining is possible, or use single-channel machining.

Programm continuation: Clear alarm with the RESET key. Restart part program

61757 **%[[Channel %1:] Block %2: %]The finished part lies outside the specified grooving limits.**

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The finished part lies outside the specified grooving limits.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check the position of the finished part contour in respect of grooving limits XDA and XDB

Programm continuation: Clear alarm with the RESET key. Restart part program

61758 **%[[Channel %1:] Block %2: %]Master spindle machine axes different in leading and following channels.**

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The machine axis references of the master spindle are different in the leading and following channels.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check machine data 35000 $MA_SPIND_ASSIGN_TO_MACHAX, 30550 $MA_AXCONF_ASSIGN_MASTER_CHAN, 20090 $MC_SPIND_DEF_MASTER_SPIND and 20070 $MC_AXCONF_MACHAX_USED

Programm continuation: Clear alarm with the RESET key. Restart part program

61759 **%[[Channel %1:] Block %2: %]Position of following channel violates software limit switch**

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: In the case of dual-channel stock removal with Balance Cutting machining (finishing or roughing with parameter DCH=0), the tool position from the leading channel from CYCLE952 is approached before the axis coupling is switched on in the following channel.
In the actual case, the tool position from the leading channel violates the software limit switch in the following channel.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: A position must be approached in the leading channel, which can also be approached in the following channel without violating the software limit switch.

Programm continuation: Clear alarm with the RESET key. Restart part program

61800 [Channel %1:] Block %2: Ext. CNC system missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Machine data for external language MD18800: $MN_MM_EXTERN_LANGUAGE or option bit 19800 $ON_EXTERN_LANGUAGE is not set.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61801 [Channel %1:] Block %2: Wrong G code selected

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: In the program call CYCLE300<value> an impermissible numerical value was programmed for the entered CNC System, or in the Cycles Setting Datum an incorrect value for the G Code System was set.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61802 [Channel %1:] Block %2: Wrong axis type

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The programmed axis is assigned to a spindle

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61803 [Channel %1:] Block %2: Programmed axis not available

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The programmed axis is not in the system.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Check parameter _AXN.
Check MD20050-20080.
Programm continuation: Clear alarm with the RESET key. Restart part program

61804 [Channel %1:] Block %2: Progr. position exceeds reference point

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The programmed intermediate position or actual position is behind the reference point.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61805 [Channel %1:] Block %2: Value programmed absolute and incremental

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The programmed intermediate position is both absolutely as well as incrementally programmed.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61806 [Channel %1:] Block %2: Wrong axis assignment

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The axis-assignment sequence is wrong.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61807 [Channel %1:] Block %2: Wrong spindle direction programmed (active)

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The programmed spindle direction contradicts the spindle direction planned for the cycle.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Check parameters SDR and SDAC.

Programm continuation: Clear alarm with the RESET key. Restart part program

61808 [Channel %1:] Block %2: Final drilling depth or single drilling depth missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The total depth Z or individual drilling depth Q is missing from theG8xblock (initial cycle call).

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61809 [Channel %1:] Block %2: Drill position not permissible

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61810 [Channel %1:] Block %2: ISO G code not possible

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: In the call block an impermissible ISO axis name was programmed.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61811 [Channel %1:] Block %2: ISO axis name illegal

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: In the call block an impermissible numerical value was programmed.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61812 [Channel %1:] Block %2: Value(s) in external cycle call wrongly defined

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: In the call block an impermissible numerical value was programmed.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61813 [Channel %1:] Block %2: GUD value wrongly defined

Explanation: An impermissible numerical value was
entered in the cycles-setting data.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61814 [Channel %1:] Block %2: Polar coordinates not possible with cycle

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: --

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61815 [Channel %1:] Block %2: G40 not active

Parameters: %1 = Channel number
%2 = Block number

Explanation: G40 was inactive before the cycle call.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: --

Programm continuation: Clear alarm with the RESET key. Restart part program

61816 [Channel %1:] Block %2: Axes not on reference point

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: --
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61817 [Channel %1:] Block %2: Axis coordinates within protection area

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: --
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61818 [Channel %1:] Block %2: Axis range limits are equal

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: --
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61819 [Channel %1:] Block %2: Risk of collision on retraction: tool violates programmed contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: With G70 in ISO mode, the contour is violated during retraction to the starting point.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Insert a G1 block at the end of the contour at the height of the starting point.
Programm continuation: Clear alarm with the RESET key. Restart part program

61840 [Channel %1:] Block %2: Error on execution of automatic servo tuning

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Acknowledgment mode
%4 = MMC sequence number

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61841 [Channel %1:] Block %2: Necessary GUD for automatic servo tuning is missing: %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Missing GUD

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61842 [Channel %1:] Block %2: Calling a cycle of the automatic servo tuning is not permissible: %3

Parameters: %1 = Channel number
%2 = Block number, label
%3 = Caller

Explanation: Calling CYCLE75x is not permissible because it is called by a cycle from the standard or manufacturer cycle directory.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: You need the manufacturer password to call CYCLE75x.

Programm continuation: Clear alarm with the RESET key. Restart part program

61850 [channel %1:] block %2: cylinder surface transformation not released

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Cylinder surface transformation not released for ShopMill.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61851 [channel %1:] block %2: no suitable transformation set up: %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The required transformation is not set up on this machine.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61852 [channel %1:] block %2: transformation not set up for this plane: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Transformation not set up for the plane used. Change plane.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61853 [channel %1:] block %2: incorrect plane for machining with rotary axis: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Machining cannot be executed on the rotary axis of the current plane. Change plane.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy:
Programm continuation: Clear alarm with the RESET key. Restart part program

61854 [Channel %1:] Block %2: Subprogram level too low for block search

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The subprogram level is too low for block search.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Execute block search with another block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61855 [Channel %1:] Block %2: Target point lies in the retraction area

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The specified target point lies in the retraction area.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Change target point or retraction area.
Programm continuation: Clear alarm with the RESET key. Restart part program

61856 [Channel %1:] Block %2: Absolute input of the work offset values not enabled

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The absolute input of the work offset value has not been enabled.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: 52212 $MCS_FUNCTION_MASK_TECH Bit 6: work offset value WO cannot be entered as absolute value (ShopTurn).
Programm continuation: Clear alarm with the RESET key. Restart part program

61857 [Channel %1:] Block %2: No rotary axis to accept a blank has been set up

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No rotary axis to accept a blank has been set up.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Check MD52207 $MCS_AXIS_USAGE_ATTRIB bit 8.
Programm continuation: Clear alarm with the RESET key. Restart part program

61858 [Channel %1:] Block %2: Turning is only possible with a blank that is clamped, centered

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Turning is only possible with a blank that is clamped, centered.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use a blank that is clamped so that it is centered.
Programm continuation: Clear alarm with the RESET key. Restart part program

61859 [Channel %1:] Block %2: Turning has not been enabled

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Turning has not been set up on the machine.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Check machine data: 52201 $MCS_TECHNOLOGY_EXTENSION=1 (turning).
Programm continuation: Clear alarm with the RESET key. Restart part program

61860 [Channel %1:] Block %2: Turning only possible on a main spindle

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: For turning, a main spindle was not selected for clamping.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select a main spindle as clamping for turning.
Programm continuation: Clear alarm with the RESET key. Restart part program

61861 [Channel %1:] Block %2: Tailstock not selected in this channel

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The tailstock was selected in another channel, but not in this channel.
The different selection/deselection results in a conflict.
When the tailstock is selected, it is not permissible that the counterspindle is positioned.
However, when the tailstock is deselected, the counterspindle should be positioned.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Select or deselect the tailstock consistently in all channels.
Programm continuation: Clear alarm with the RESET key. Restart part program

61862 [Channel %1:] Block %2: It is not permissible to call a ShopTurn cycle in a ShopMill program

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: An attempt was made to call a ShopTurn cycle in a ShopMill program.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Delete block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61863 [Channel %1:] Block %2: It is not permissible to call a ShopMill cycle in a ShopTurn program

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: An attempt was made to call a ShopMill cycle in a ShopTurn program.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Delete block.
Programm continuation: Clear alarm with the RESET key. Restart part program

61864 [Channel %1:] Block %2: The selection, tailstock yes/no must be identical in all channels

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The selection, tailstock yes/no in the program header must be identical in all channels.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: For tailstock yes/no in the program header, make the same selection for all channels.
Programm continuation: Clear alarm with the RESET key. Restart part program

61865 [Channel %1:] Block %2: Calling a ShopTurn cycle is only permissible in a ShopTurn program

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: It is not permissible that a ShopTurn cycle is used outside a ShopTurn program, as the required environment variables are otherwise not assigned.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Calling a ShopTurn cycle in a ShopTurn program. If this is not possible, then the task must be programmed using G code.
Programm continuation: Clear alarm with the RESET key. Restart part program

61866 [Channel %1:] Block %2: It is only permissible to call a ShopMill cycle in a ShopMill program

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: It is not permissible that a ShopMill cycle is used outside a ShopMill program, as the required environment variables are otherwise not assigned.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Calling a ShopMill cycle in a ShopMill program. If this is not possible, then the task must be programmed using G code.
Programm continuation: Clear alarm with the RESET key. Restart part program

61867 [Channel %1:] Block %2: On face C, the retraction distance must not be larger than the distance to the center point.

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Retraction distance too large. On face C, the retraction distance must not be larger than the distance to the center point. Otherwise, retraction would become negative, which would lead to a reversal of the work spindle.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use a smaller retraction distance or face Y if possible.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61868 [Channel %1:] Block %2: Program was created on another machine and must be adjusted.

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: The program was created on another machine and contains program parts that can not be executed on this machine.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: The program must be adjusted in the ShopMill/ShopTurn editor.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61869 [Channel %1:] Block %2: Block search is only possible with calculation

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: With ShopMill and ShopTurn, block search is only possible with calculation.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Use a different block search mode.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61870 [Channel %1:] Block %2: Danger of collision during automatic retraction. Retract tool manually!

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: There is a danger of collision during automatic retraction.
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: Retract tool manually!
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61900 %[[Channel %1:] Block %2: %]Name of program to be generated is missing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check parameter PRG
Programm continuation: Clear alarm with the RESET key. Restart part program

61901 %[[Channel %1:] Block %2: %]Contour %4 does not exist

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check contour call
- Check whether the contours exist in the program storage (workpieces, subroutines or part programs)
Programm continuation: Clear alarm with the RESET key. Restart part program

61902 %[[Channel %1:] Block %2: %]Label %4 not existing in the pocket contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check whether the labels exist in the pocket contour
Programm continuation: Clear alarm with the RESET key. Restart part program

61903 %[[Channel %1:] Block %2: %]Label %4 not existing in the blank contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check whether the labels exist in the blank contour
Programm continuation: Clear alarm with the RESET key. Restart part program

61904 %[[Channel %1:] Block %2: %]Label %4 not existing in the island contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check whether the labels exist in the island contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61905 %[[Channel %1:] Block %2: %]Label %4 not existing in the spigot contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check whether the labels exist in the spigot contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61906 %[[Channel %1:] Block %2: %]Label %4 not existing in the contour

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check whether the labels exist in the contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61907 %[[Channel %1:] Block %2: %]Pocket contour missing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check contour call
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61908 %[[Channel %1:] Block %2: %]Blank contour missing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check contour call
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61909 %[[Channel %1:] Block %2: %]Error in pocket contour %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the pocket contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61910 %[[Channel %1:] Block %2: %]Error in the blank contour %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the blank contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61911 %[[Channel %1:] Block %2: %]Error in island contour %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the island contour
- Check whether the island contour is closed
- Check the contour for self-cuts
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61912 %[[Channel %1:] Block %2: %]Error in spigot contour %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the spigot contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61913 %[[Channel %1:] Block %2: %]Error in contour %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the contour
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61914 %[[Channel %1:] Block %2: %]Too many contours specified

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check the number of contours
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61915 %[[Channel %1:] Block %2: %]Cutter radius too small

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check the radius of the milling cutter in tool management
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61916 %[[Channel %1:] Block %2: %]Calculation has been canceled

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Calculation has been canceled by PI service; please try again
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61917 %[[Channel %1:] Block %2: %]Combination of centering/predrilling and spigot not allowed

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Machining of spigot in conjunction with predrilling/centering not allowed!
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61918 %[[Channel %1:] Block %2: %]Cut. radius for residual mach. must be smaller than cut. radius for ref. tool

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check cutter radius for residual machining which must be smaller than cutter radius for reference tool !
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61919 %[[Channel %1:] block %2: %]Radius of the reference tool is too small

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check the radius of the reference tool.
Programm continuation: Clear alarm with the RESET key. Restart part program

---

61920 %[[Channel %1:] Block %2: %]Error during contour milling %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: 3020: No more memory is available
3022: The contour is incorrect
3023: Check contour (contour may contain too small elements)
3356: The residual material cannot be removed completely with the tool used.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: 3020, 3022, and 3023: Check contour and technological data
3356: A tool with a smaller diameter, with which the residual material can be removed completely, must be used for residual machining.
Call the SIEMENS hotline

Programm continuation: Clear alarm with the RESET key. Restart part program

61921 %[[Channel %1:] Block %2: %]Error during groove milling %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:

Programm continuation: Clear alarm with the RESET key. Restart part program

61922 %[[Channel %1:] Block %2: %]Label %4 not present in the groove contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check whether the labels are present in the groove contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61923 %[[Channel %1:] Block %2: %]Error in the groove contour %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check the programming of the groove contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61924 %[[Channel %1:] Block %2: %]Groove contour missing

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contour call

Programm continuation: Clear alarm with the RESET key. Restart part program

61930 %[[Channel %1:] Block %2: %]No contour available

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contour call
- Check whether the contours exist in the program storage (workpieces, subroutines or part programs)

Programm continuation: Clear alarm with the RESET key. Restart part program

61931 %[[Channel %1:] Block %2: %]Contour not closed

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check whether the contours are closed

Programm continuation: Clear alarm with the RESET key. Restart part program

61932 %[[Channel %1:] Block %2: %]Selfcutting contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Modify contour programming

Programm continuation: Clear alarm with the RESET key. Restart part program

61933 %[[Channel %1:] Block %2: %]Too many contour elements

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Modify contour programming and thereby try to reduce the number of contour elements

Programm continuation: Clear alarm with the RESET key. Restart part program

61934 %[[Channel %1:] Block %2: %]Programming of the machining plane not allowed here

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Modify contour programming

Programm continuation: Clear alarm with the RESET key. Restart part program

61935 %[[Channel %1:] Block %2: %]Programming of inch/metric measuring system not allowed here

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Modify contour programming

Programm continuation: Clear alarm with the RESET key. Restart part program

61936 %[[Channel %1:] Block %2: %]G0 is not allowed in contour programming

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Modify contour programming, replace G0 by G1

Programm continuation: Clear alarm with the RESET key. Restart part program

61937 %[[Channel %1:] Block %2: %]Pocket depth programmed incorrectly

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter Z1

Programm continuation: Clear alarm with the RESET key. Restart part program

61938 %[[Channel %1:] Block %2: %]No starting point specified

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter for specified starting point,
- for G17: XS, YS
- for G18: ZS, XS
- for G19: YS, ZS

Programm continuation: Clear alarm with the RESET key. Restart part program

61939 %[[Channel %1:] Block %2: %]No center point specified for circle

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check contour programming, particularly circle programming

Programm continuation: Clear alarm with the RESET key. Restart part program

61940 %[[Channel %1:] Block %2: %]Specified starting point programmed incorrectly

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Correct specified starting point

Programm continuation: Clear alarm with the RESET key. Restart part program

61941 %[[Channel %1:] Block %2: %]Helix radius too small

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Increase helix radius

Programm continuation: Clear alarm with the RESET key. Restart part program

61942 %[[Channel %1:] Block %2: %]Helix violates contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check helix radius and reduce in size, if possible

Programm continuation: Clear alarm with the RESET key. Restart part program

61943 %[[Channel %1:] Block %2: %]Approach/retract motion violates contour

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Reduce safety clearance SC, if possible

Programm continuation: Clear alarm with the RESET key. Restart part program

61944 %[[Channel %1:] Block %2: %]Ramp path too short

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: During oscillating insertion, the message "Ramp path too short" occurs if the tool moves less than the mill diameter away from the insertion point on the ramp path or the machining depth is not reached.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Reduce the insertion angle if the tool remains too near to the insertion point
- Enlarge insertion angle if the tool does not reach the machining depth
- Use tool with smaller radius
- Use another insertion mode

Programm continuation: Clear alarm with the RESET key. Restart part program

61945 %[[Channel %1:] Block %2: %]Plane infeed too large, residual corners remaining

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check parameter for plane infeed
- for G17: DXY
- for G18: DZX
- for G19: DYZ

Programm continuation: Clear alarm with the RESET key. Restart part program

61946 %[[Channel %1:] Block %2: %]Island contour existing twice

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Delete double island contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61947 %[[Channel %1:] Block %2: %]Spigot contour existing twice

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Delete double spigot contour

Programm continuation: Clear alarm with the RESET key. Restart part program

61948 %[[Channel %1:] Block %2: %]No material available for machining

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check programming of the contours
Programm continuation: Clear alarm with the RESET key. Restart part program

61949 %[[Channel %1:] Block %2: %]Island is outside the pocket

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check programming of the island/pocket contour
Programm continuation: Clear alarm with the RESET key. Restart part program

61950 %[[Channel %1:] Block %2: %]No residual material available

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: --
Programm continuation: Clear alarm with the RESET key. Restart part program

61951 %[[Channel %1:] Block %2: %]Cutter radius for residual material too large

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Use cutter with smaller radius
Programm continuation: Clear alarm with the RESET key. Restart part program

61952 %[[Channel %1:] Block %2: %]Radius of res. material cutter too small in relation to ref. cutter

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Use a cutter with a larger radius for residual machining

Programm continuation: Clear alarm with the RESET key. Restart part program

61953 %[[Channel %1:] Block %2: %]Technology change to preprocessing not possible

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: - Technology of the current machining does not match the technology of the preprocessing

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Adapt the technology of the current machining to the technology of the preprocessing

Programm continuation: Clear alarm with the RESET key. Restart part program

61954 %[[Channel %1:] Block %2: %]Active and programmed tools are different

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: - Active and programmed tools are different

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check active and programmed tools

Programm continuation: Clear alarm with the RESET key. Restart part program

61955 %[[Channel %1:] Block %2: %]Internal memory limit reached for contour calculation

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: - The internal memory is insufficient to calculate the contour.

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - Check the following parameters/possibilities:
- The angle of contact can be increased
- A tool with a larger diameter can be used
- Can be reset in setting data $SCS_FUNCTION_MASK_MILL_SET bit3

Programm continuation: Clear alarm with the RESET key. Restart part program

61956 %[[Channel %1:] Block %2: %]Active tool radius must be less than or equal to the radius of the reference tool

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Interpreter stop
NC Start disable in this channel.
Interface signals are set.
Alarm display.
Remedy: - Check the active tool radius, this must be less than or equal to the radius of the reference tool
Programm continuation: Clear alarm with the RESET key. Restart part program

62000 [Channel %1:] Block %2: Insert new tool

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Please load new tool.
Reaction: Alarm display.
Remedy: --
Programm continuation: Clear alarm with the Delete key or NC START.

62098 [Channel %1:] Block %2: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: This alarm is used for various purposes.
Please refer to the alarm text.
Reaction: Alarm display.
Remedy: Depends on alarm text
Programm continuation: Clear alarm with the Delete key or NC START.

62100 [Channel %1:] Block %2: No drilling cycle active

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No modal drilling cycle has been called before the drilling pattern cycle call.
Reaction: Alarm display.
Remedy: Check whether a drilling cycle was called prior to calling the drilling pattern cycle.
Programm continuation: Clear alarm with the Delete key or NC START.

62101 [Channel %1:] Block %2: Milling direction incorrect - G3 is generated

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Synchronous or reverse rotation programmed. But the spindle does not rotate at a cycle call.
Reaction: Alarm display.
Remedy: Check value in paramter CDIR.
Programm continuation: Clear alarm with the Delete key or NC START.

62102 [Channel %1:] Block %2: pocket not completely solidly machined during finishing

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Alarm display.

Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

62103 [Channel %1:] Block %2: No finishing allowance programmed

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: No finishing allowance is programmed, although it is necessary for this machining.
Reaction: Alarm display.
Remedy: Programm a finishing allowance.
Programm continuation: Clear alarm with the Delete key or NC START.

62104 [Channel %1:] Block %2: Drilling cycle incorrectly defined

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Alarm display.
Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

62105 [Channel %1:] Block %2: Number of columns or lines equals zero

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Alarm display.
Remedy: Check parameters _NUM1 and _NUM2.
Programm continuation: Clear alarm with the Delete key or NC START.

62106 [Channel %1:] Block %2: incorrect value for monitoring status in tool monitoring

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Alarm display.
Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

62107 [Channel %1:] Block %2: parameter %4 incorrectly defined for tool monitoring in cycles

Parameters: %1 = Channel number
%2 = Block number, label
Explanation:
Reaction: Alarm display.
Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

62108 [Channel %1:] Block %2: error in function Tool monitoring in cycles

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Alarm display.

Remedy:

Programm continuation: Clear alarm with the Delete key or NC START.

62180 [Channel %1:] Block %2: Set rotary axes %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Sample display of the swivel angles to be set for manual rotary axes:
62180 "Set rotary axes B=32.5° C=45°"

Reaction: Alarm display.

Remedy: Settable angles for manual rotary axes.

Programm continuation: Clear alarm with the Delete key or NC START.

62181 [Channel %1:] Block %2: Set rotary axis %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: Sample display of the swivel angle to be set for a manual rotary axis:
62181 "Set rotary axis B=32.5°"

Reaction: Alarm display.

Remedy: Settable angle for manual rotary axis.

Programm continuation: Clear alarm with the Delete key or NC START.

62182 [Channel %1:] Block %2 : load swivel head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Alarm display.

Remedy: Request to load a swivel head.

Programm continuation: Clear alarm with the Delete key or NC START.

62183 [Channel %1:] Block %2 : unload swivel head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:

Reaction: Alarm display.

Remedy: --

Programm continuation: Clear alarm with the Delete key or NC START.

62184 [Channel %1:] Block %2 : replace swivel head: %4

Parameters: %1 = Channel number
%2 = Block number, label

Explanation:
Reaction: Alarm display.
Remedy: --
Programm continuation: Clear alarm with the Delete key or NC START.

62185 [Channel %1:] Block %2 : angle adapted to angle grid: %4

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: %4 difference angle with Hirth tooth system
Reaction: Alarm display.
Remedy: Check setup of swivel CYCLE800.
Programm continuation: Clear alarm with the Delete key or NC START.

62186 [Channel %1:] Block %2: No swiveling in JOG -> WO G%4 active and total basic WO (G500) contain rotations

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: On swiveling in JOG no rotation can be written to the work offset WO,
if rotations are already contained in the total basic WO or in the basic reference
Error 62186 message can be masked -> see setting data 55410 $SCS_MILL_SWIVEL_ALARM_MASK
Reaction: Alarm display.
Remedy: %4 number of the active work offset WO.
Programm continuation: Clear alarm with the Delete key or NC START.

62187 [Channel %1:] Block %2: Swiveling in JOG - G500 active and total basic WO or basic reference contain rotations

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: During swiveling in JOG, it is not possible to write a rotation into the work offset WO,
if, with active G500, rotations are already contained in the total basic WO or in the basic reference
Error message 62187 can be masked -> see setting data 55410 $SCS_MILL_SWIVEL_ALARM_MASK
Reaction: Alarm display.
Remedy: See notes for 62186 and 62187.
Programm continuation: Clear alarm with the Delete key or NC START.

62200 [Channel %1:] Block %2: Start spindle

Parameters: %1 = Channel number
%2 = Block number, label
Explanation: Stop prior to thread machining, as the spindle is in stop position.
Reaction: Alarm display.
Remedy: Start the tool spindle before machining the thread.
Programm continuation: Clear alarm with the Delete key or NC START.

62201 [Channel %1:] Block %2: Z offset does not influence the retraction planes.

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: The retraction planes refer to the workpiece. Therefore, programmable offsets do not influence the retraction planes.

Reaction: Alarm display.

Remedy: Check that offset does not cause collision.
Then press NC Start.

Programm continuation: Clear alarm with the Delete key or NC START.

62202 [Channel %1:] Block %2: NOTICE: tool travels directly to machining!

Parameters: %1 = Channel number
%2 = Block number, label

Explanation: After block search a position is to be reached by direct approach.

Reaction: Alarm display.

Remedy: Check whether the desired position can be reached without collision.
Then execute an NC start.

Programm continuation: Clear alarm with the Delete key or NC START.

62300 [Channel %1:] Block %2: Check number of empirical value memory

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: --

Reaction: Alarm display.

Remedy: Check setpoint value

Programm continuation: Clear alarm with the Delete key or NC START.

62301 [Channel %1:] Block %2: Notice! Search run, test run or simulation active

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: - Deactivate program test or test run

Programm continuation: Clear alarm with the Delete key or NC START.

62303 [Channel %1:] Block %2: Safety margin exceeded %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: - Check setpoint value and parameter _TSA

Programm continuation: Clear alarm with the Delete key or NC START.

62304 [Channel %1:] Block %2: Allowance

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The difference between actual and setpoint measurements is greater than the upper tolerance limit (screen parameter TUL):
- TUL is the upper tolerance limit of the measurement difference.
- TUL is always related to the material, irrespective of whether external or internal machining is involved.
- The hole/pocket is thus too small and the spigot is too large.
- This means that further material can be removed.
- The measuring cycle parameter TUL corresponds to "upper deviation", the usual term used in mechanical engineering for fits and tolerances.

Reaction: Alarm display.

Remedy: The difference between actual and setpoint value is larger than upper tolerance limit (parameter _TUL).

Programm continuation: Clear alarm with the Delete key or NC START.

62305 [Channel %1:] Block %2: Dimension too small

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: The difference between the actual and setpoint measurement is less than the lower tolerance limit (screen parameter TLL):
- TLL is the lower tolerance limit of the measurement difference.
- TLL is always related to the material, irrespective of whether external or internal machining is involved.
- The hole/pocket is thus too large and the spigot too small.
- This means that too much material has already been removed.
- The measuring cycle parameter TLL corresponds to "lower deviation", the usual term used in mechanical engineering for fits and tolerances.

Reaction: Alarm display.

Remedy: The difference between actual and setpoint value is smaller than lower tolerance limit (parameter _TLL).

Programm continuation: Clear alarm with the Delete key or NC START.

62306 [Channel %1:] Block %2: Permissible measuring difference exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: The difference between actual and setpoint value is larger than tolerance parameter _TDIF, tool data are not corrected.

Programm continuation: Clear alarm with the Delete key or NC START.

62307 [Channel %1:] Block %2: Maximum number of characters per line exceeded.

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: Insufficient number of characters per line.

Reaction: Alarm display.

Remedy: Increase the value in _PROTFORM[1]

Programm continuation: Clear alarm with the Delete key or NC START.

62310 [Channel %1:] Block %2: The max. number of characters per line is limited to %4 characters per line

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:
Reaction: Alarm display.
Remedy: --
Programm continuation: Clear alarm with the Delete key or NC START.

62311 [Channel %1:] Block %2: The maximum number of characters per line _PROTFORM[1] is adjusted.

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation: Max. number of characters per line _PROTFORM[1] has been adjusted.
Reaction: Alarm display.
Remedy: --
Programm continuation: Clear alarm with the Delete key or NC START.

62312 [Channel %1:] Block %2: probe is not perpendicular to plane!

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Reaction: Alarm display.
Remedy: --
Programm continuation: Clear alarm with the Delete key or NC START.

62314 [Channel %1:] Block %2: Path limitation via software end position %4, collision detection activated, continue with NC START / cancel with RESET

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation: This alarm is only output with external measurements (web, spigot, ball) as a query about the program continuation before the lowering movement in the infeed axis
The alarm scan is caused by the reduction of the path of the measuring axis in the working plane that has just been made.
If the path had not been reduced, the traversing movement would have extended beyond the software end position of the working plane. A corresponding NCK alarm 10722 would have been triggered.
With program stop, the user can check whether the infeed axis can be lowered to measuring height without causing a collision.
Alarm 61402 is output if there is a switch (collision) of the probe in the subsequent movement of the infeed axis.
Reaction: Alarm display.
Remedy: - Check setpoint value specification.
- Place the workpiece further away from the software limit position.
Programm continuation: Clear alarm with the Delete key or NC START.

62315 [Channel %1:] Block %2: Overwrite kinematics data record %4, yes -> NC start, no -> reset

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Reaction: Alarm display.
Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

62316 [Channel %1:] Block %2: Overwrite TRAORIdata, yes -> NC start, no -> reset

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy:

Programm continuation: Clear alarm with the Delete key or NC START.

62317 [Channel %1:] Block %2: Tolerance of the linear vector %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: If the error comes from CYCLE996 or CYCLE9960:
The tolerance of the linear vectors is exceeded during the measurement of the kinematics.
The value of the tolerance is transferred in parameter TLIN.
If TLIN=0 or tolerance (check) = no, the measured vectors are not monitored.

Reaction: Alarm display.

Remedy: A 2nd measurement with a larger tolerance can be made for the analysis. The kinematic data should not be overwritten.
The newly measured vectors are documented in the measurement log (data file).

Programm continuation: Clear alarm with the Delete key or NC START.

62318 [Channel %1:] Block %2: Tolerance of the rotary axis vector %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy:

Programm continuation: Clear alarm with the Delete key or NC START.

62319 [Channel %1:] Block %2: No internal correction of the calibration data

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: Check probe alignment/spindle position!
The alignment (programmed position) of the tool probe in the workspindle must be identical during calibration and measuring!
If these positions vary, the calibration data cannot be corrected cycle-internally with regards to a coordinate rotation of the workplane around the infeed axis!

Programm continuation: Clear alarm with the Delete key or NC START.

62320 [Channel %1:] Block %2 Individual edges lie outside the dimensional difference: %4

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: Measurement of the individual edges shows that the specified number of edges lies outside the dimensional difference.
A decision has to be made: whether or not one can continue working with this tool.

Programm continuation: Clear alarm with the Delete key or NC START.

62321 [Channel %1:] Block %2: Rotary axis 1: Diameter tolerance of calibration ball between measurement %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: see measured diameter in the parameters _OVR[72] to _OVR[74] or the value in SD $SCS_MEA_KIN_DM_TOL

Reaction: Alarm display.

Remedy: Check the calibration data or recalibrate the 3D probe
Check the mechanical structure of the calibration ball in the machine

Programm continuation: Clear alarm with the Delete key or NC START.

62322 [Channel %1:] Block %2: Rotary axis 2: Diameter tolerance of calibration ball between measurement %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation: see measured diameter in the parameters _OVR[75] to _OVR[77] or the value in SD $SCS_MEA_KIN_DM_TOL

Reaction: Alarm display.

Remedy: Check the calibration data or recalibrate the 3D probe
Check the mechanical structure of the calibration ball in the machine

Programm continuation: Clear alarm with the Delete key or NC START.

62377 [Channel %1:] Block %2: Dimensional tolerance %4 exceeded

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy:

Programm continuation: Clear alarm with the Delete key or NC START.

62500 [Channel %1:] Block %2: GWPS has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: Check limit value for GWPS and program a lower value in the NC program if necessary

Programm continuation: Clear alarm with the Delete key or NC START.

62501 [Channel %1:] Block %2: Speed has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number

Explanation:

Reaction: Alarm display.

Remedy: Check limit value for speed and program a lower value in the NC program if necessary
Programm continuation: Clear alarm with the Delete key or NC START.

62502 [Channel %1:] Block %2: Dresser %4, GWPS has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Reaction: Alarm display.
Remedy: Check limit value for GWPS and program a lower value in the NC program if necessary
Programm continuation: Clear alarm with the Delete key or NC START.

62503 [Channel %1:] Block %2: Dresser %4, speed has been limited

Parameters: %1 = Channel number
%2 = Block number, label channel number
Explanation:
Reaction: Alarm display.
Remedy: Check limit value for speed and program a lower value in the NC program if necessary
Programm continuation: Clear alarm with the Delete key or NC START.

5.4 Drive alarms

To view detailed description about individual alarms directly on the PPU, proceed as follows:

1. Press this key on the PPU to enter the alarm operating area.

2. Select the desired alarm.

3. Press this key to open the online help for the selected alarm.

Note: You can further press this softkey in the current help screen to show a complete list of all SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms. In addition, you can also use the following softkey to search for a specific alarm by number in this list:

4. Pressing this softkey exits the help system.

300402 System error in drive link. Error codes %1, %2

Parameters: %1 = Error code 1
%2 = Error code 2
Explanation: An internal software error or serious error condition has occurred, which could possibly be rectified by a hardware reset.

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: If you experience such a system error, please contact Technical Support.
www.siemens.com/sinumerik/help
Please supply the following information to ensure quick processing:
- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>

Programm continuation: Switch control OFF - ON.

300406 Problem in the non-cyclic communication for basic address %1, additional information %2, %3, %4

Explanation: For PROFIdrive only:
A problem occured during the non-cyclic communication with the logical start address. The additional information defines the location of the problem. If the logical start address 0 is output, only the additional information is relevant.

Reaction: Alarm display.
Warning display.

Remedy: Please inform the authorized personnel/service department. The alarm can be suppressed with MD11411 $MN_ENABLE_ALARM_MASK bit 1 = 0
If you experience such a system error, please contact Technical Support.
www.siemens.com/sinumerik/help
Please supply the following information to ensure quick processing:
- Alarm number together with alarm text
- Description of the operation/mode before the alarm message
- Generate log files using the key combination: <Ctrl> + <Alt> + <D>

Programm continuation: Clear alarm with the Delete key or NC START.

300410 Axis %1 drive %2 error when storing a file (%3, %4)

Parameters: %1 = NC axis number
%2 = Drive number
%3 = Error code 1
%4 = Error code 2

Explanation: An attempt to save a data block, e. g. the result of a measuring function, in the file system has failed.
On error code 1 == 291: An error occurred during preparation of the ACC information. Basic information prepared on the drive contains an error or has an unknown format.
On error code 1 == 292: Memory shortage during preparation of the ACC information.

Reaction: Interface signals are set.
Alarm display.

Remedy: - Please inform the authorized personnel/service department.
- Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself (this will require a complete data backup).
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18321 $MM_MAXNUM_SYSTEM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of
- 18270 $MM_NUM_SUBDIR_PER_DIR,
- 18310 $MM_NUM_DIR_IN_FILESYSTEM,
- Power On
- Reload saved data
- On error code 1 == 291: Replace the drive software and use version with suitable ACC basic information.
- On error code 1 == 292: Replace the drive software and use fewer different versions of the drive software.

Programm continuation: Clear alarm with the RESET key. Restart part program

300412 Error when storing a file (%1, %2)

Parameters: %1 = Error code 1
%2 = Error code 2

Explanation: An attempt to save a data block, e.g. the result of a measuring function, in the file system has failed.

Reaction: Interface signals are set.
Alarm display.

Remedy: Please inform the authorized personnel/service department. Create more space in the file system. It is normally sufficient to delete 2 NC programs or to free 4 - 8 Kbytes of memory. If these remedies do not work, it will be necessary to increase the number of files per directory or the size of the file system itself. To do so, proceed as follows:
- Save all data
- Change settings of machine data
- 18280 $MM_NUM_FILES_PER_DIR
- 18320 $MM_NUM_FILES_IN_FILESYSTEM
- 18321 $MM_MAXNUM_SYSTEM_FILES_IN_FILESYSTEM
- 18350 $MM_USER_FILE_MEM_MINIMUM
- and, if necessary, of
- 18270 $MM_NUM_SUBDIR_PER_DIR
- 18310 $MM_NUM_DIR_IN_FILESYSTEM
- Power On
- Reload saved data

Programm continuation: Clear alarm with the RESET key. Restart part program

300423 Measuring result could not be read (%1)

Parameters: %1 = Error code

Explanation: An attempt to read a measurement result has failed:
- Error code = 4: Not enough space for test result
- Error code = 16: Measurement not yet finished

Reaction: Interface signals are set.
Alarm display.

Remedy: Repeat measurement. Alter measuring time if necessary.

Programm continuation: Clear alarm with the RESET key. Restart part program

380001 PROFIBUS/PROFINET: Startup error, reason %1 parameter %2 %3 %4.

Parameters: %1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Explanation: An error occurred during startup of the PROFIBUS/PROFINET master.
Overview: Cause of the error, Par 1, Par 2, Par 3:
- 01 = DPM version, DPM version, DPA version, --
- 02 = DPM ramp-up timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM ramp-up status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM ramp-up error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = DPM-PLL sync error, --, --, --
- 07 = Alarm queue too long, Actual number, Setpoint number, --
- 08 = Unknown client, Client ID, --, --
- 09 = Client version, Client ID, Client version, DPA version
- 10 = Too many clients, Client number, max. number of clients, --
- 11 = Log.basic address used several times; bus no.; slot no.; log.basic address --
- 20 = Slave/device address used several times, slave/device address --
- 21 = Slave/device address unknown, slave/device address, --
- 22 = Erroneous configuration telegram, slave/device address, error code, --
- 23 = OMI incompatible (data), drive version, CDA version, --, --
- 24 = OMI incompatible (driver), drive version, CDA version, --, --
- 25 = CPI initialization failed, error code, --, --, --
- 26 = DMA not active
- 27 = Reserved
- 28 = Reserved
- 29 = Reserved
- The 1000s digit of the error cause = number of the affected bus
- (Special case: Error causes in the 5000 range indicate problems with the NCU-LINK communication)

Clients are the following components of the control system that use the PROFIBUS/PROFINET:
Client ID = 1: PLC
Client ID = 2: NCK
Possible causes are:
- Error in contents of SDB
- Corruption of parts of the system program
- Hardware defect on NC component

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:
Remedy for 1-11
1. Check the control project, check MD11240 $MN_PROFIBUS_SDB_NUMBER, and reload it when using a user-specific SDB.
2. Increase the monitoring time for the PLC run-up in MD10120 $MN_PLC_RUNNINGUP_TIMEOUT
3. If the error still occurs, back up data, and restart the control with the standard values as per the as-delivered condition.
4. If run-up is satisfactory, reload the user data stage by stage.
5. If the error still occurs during run-up with standard values, reboot the system from the PC card or update the software.
6. If the error still occurs, replace the hardware.
Remedy for 20-21
1. Check/correct the addresses of the connected slaves/devices.
Remedy for 22
See SINAMICS warning 1903 for a description of the meaning behind the error codes.
1. Check the SDB
- Check the type and length of the message frame
- Match slot assignment with P978
2. Evaluate the drive alarms/warnings
Remedy for 23-24
1. Software replacement required
Remedy for 25
1. Change the message frame type
2. Reduce the number of slots
3. Reduce the number of slaves/devices
4. Create a new SDB
5. Software must be replaced
If the error can still not be rectified after this procedure, send the error message to the control manufacturer.

Programm continuation:
Switch control OFF - ON.

380003 PROFIBUS/PROFINET: Operating error, reason %1 parameter %2 %3 %4.

Parameters:
%1 = Cause of the error
%2 = Parameter 1
%3 = Parameter 2
%4 = Parameter 3

Explanation:
An operating error occurred on the PROFIBUS/PROFINET in cyclic mode.
Overview: Cause of the error, Par 1, Par 2, Par 3:
- 01 = unknown alarm, alarm class, logical address, --
- 02 = DPM cycle timeout, DPM actual value status, DPM setpoint value status, --
- 03 = DPM cycle status, DPM actual value status, DPM setpoint value status, DPM error code
- 04 = DPM cycle error, DPM actual value status, DPM setpoint value status, DPM error code
- 05 = Client not registered, client number, max. number of clients, --
- 06 = Synchronization error, number of sync violation, --, --
- 07 = Spinlock timeout, PLC spinlock, NCK spinlock, --
- 1000s digit of the error cause = number of the affected bus
- (Special case: Error causes in the 5000 range indicate problems with the NCU-LINK communication)
Alarm class: (see alarm 380 060)
The following can be primary causes:
- For error cause 01: Data transfer error on the PROFIBUS/PROFINET
- For error causes 02, 03, 04: Error in contents of SDB
- For error causes 02, 03, 04, 05, 07: Corruption of parts of system program
- For error cause 06: The PCI bus cycle does not match the expected rate, so synchronization is not possible. The correct PCI bus cycle must be entered.
The error can also be caused by a hardware problem on the MCI module.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: - For error cause 01:
- Check the electrical and fault-related specifications for PROFIBUS/PROFINET, assess the cable installation
- Check the terminating resistors of the PROFIBUS connectors (ON setting at ends of cables, otherwise OFF setting required)
- Check slave/device
- For error causes 02, 03, 04:
- Check SDB
- For error causes 02, 03, 04, 05, 07:
- Follow the procedure described for troubleshooting alarm 380 001
- For error cause 06:
- The correct PCI bus cycle must be entered.
If the error cannot be eliminated by this procedure, send the error text to the control system manufacturer.

Programm continuation: Clear alarm with the RESET key. Restart part program

380005 PROFIBUS/PROFINET: Bus %3 access conflict, type %1, counter %2

Parameters: %1 = Conflict type
%2 = Serial number within a conflict sequence
%3 = Number of the affected bus

Explanation: An access conflict occurred on the PROFIBUS/PROFINET in cyclic mode: An attempt has been made in the NCK to write data to the bus or to read from the bus while cyclic data transfer was active. This may lead to inconsistant data.
Type 1: NCK attempts to read data before the cyclic transfer has finished on the bus.
Type 2: The NCK has not finished writing its data when the cyclic transfer begins again. Counter %2 contains a serial number starting at 1. A maximum of 10 alarms are output in succession. If no conflicts occur in a DP cycle, the counter is reset and new alarms are output again on the next conflict.

Reaction: Alarm display.

Remedy: - Check the timing again, in particular ensure that the settings in MD10050 $MN_SYSCLOCK_CYCLE_TIME and MD10062 $MN_POSCTRL_CYCLE_DELAY are correct:
MD10062 $MN_POSCTRL_CYCLE_DELAY must be larger for type 1.
MD10062 $MN_POSCTRL_CYCLE_DELAY must be smaller for type 2.
- If alarm-free operation cannot be achieved with any MD10062 $MN_POSCTRL_CYCLE_DELAY setting, MD10050 $MN_SYSCLOCK_CYCLE_TIME must be increased.
- If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Programm continuation: Clear alarm with the Delete key or NC START.

380020 PROFIBUS/PROFINET: Bus %3 SDB %4 error %1 source %2

Parameters: %1 = Error
%2 = SDB source
%3 = Bus number
%4 = SDB number

Explanation: Error in SDB for configuring PROFIBUS/PROFINET.
Causes of the error:
- 01 = SDB does not exist in source.
- 02 = SDB from source is too large.
- 03 = SDB from source cannot be activated.
- 04 = Source is empty.
- 05 = Source is not present.

SDB source:
- 99 = Passive file system: _N_SDB_DIR
- 100 = CF card: /siemens/sinumerik/sdb/...
- 101 = CF card: /addon/sinumerik/sdb/...
- 102 = CF card: /oem/sinumerik/sdb/...
- 103 = CF card: /user/sinumerik/sdb/...

Reaction: PROFIBUS/PROFINET is inactive or working with the default SDB.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy:
- Check the setting of MD 11240 $MN_PROFIBUS_SDB_NUMBER.
- If source = 100: Check directory _N_SDB_DIR in the passive file system.
- If source = 103-106: Check directories on CF card

Programm continuation: Switch control OFF - ON.

380022 PROFIBUS/PROFINET: Configuration of DP master bus %1 has been changed

Parameters: %1 = Number of the affected bus

Explanation: The PROFIBUS configuration on the DP master was changed during operation, e.g. by downloading a new hardware configuration via STEP 7. As the cycle data may also have changed, operation cannot be continued, and a warm start is required.
If the master functionality is within the PLC (as on the 840Di), the PLC will have been stopped for the download, and alarm 2000 (PLC sign-of-life) output.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Programm continuation: Switch control OFF - ON.

380040 PROFIBUS/PROFINET: Bus %3, configuration error %1, parameter %2

Parameters: %1 = Cause of the error
%2 = Parameter
%3 = Number of the affected bus

Explanation: The generation of the PROFIBUS/PROFINET in the SDB does not conform to the configuration specifications of the NC in use.

Overview: Cause of the error, par 1:

- 01 = SDB contains slave/device without diagnostics slot, slave/device address
- 02 = SDB contains too many slot entries, identifier
- 03 = SDB contains no equidistance data, no function
- 04 = PNIO: SDB contains different Tdp (also TDC) on one device
- 05 = PNIO: SDB contains different Tmapc (also CACF) on one device
- 06 = PNIO: SDB contains different TI on one device
- 07 = PNIO: SDB contains different TO on one device
- 08 = PNIO: SDB contains device numbers that are too high (with values higher than 126)
- 09 = SDB content transferred segmented (too many slots/frames)
- 10 = Not enough memory space for segmented SDB content (too many slots/frames)
- 11 = The telegram configured in the SDB is too short for the selected telegram acc. to $MN_DRIVE_TELEGRAM_TYPE.
- 20 = SDB contains too many slaves/devices, quantity
- 21 = SDB missing or contains invalid data, error code
- 22 = SDB configuration data incorrect, slave/device address, error code
- 23 = Reserved
- 24 = Reserved
- 25 = Reserved
- 26 = Reserved
- 27 = Reserved
- 28 = Reserved
- 29 = Reserved

Reaction: Channel not ready.

NC Start disable in this channel.

Interface signals are set.

Alarm display.

Remedy: Check that the corresponding SDB:

- Contains a diagnostic slot for every slave/device
- Contains only slave/device entries relevant to the application

In principle, it is possible to include a superset of slaves/devices in the SDB that are partially relevant for different end versions of the product. However, this overloads the NC memory and runtime capacity and should, therefore, always be avoided.

If this alarm occurs, reduce the SDB to a minimum.

If the code for the error cause is 03, check that equidistance is activated in the SDB (e.g. using STEP 7 HW config).

If the code for the error cause is 10, reduce the number of slaves/slots on the bus concerned (e.g. using STEP 7 HW config).

If the alarm continues to occur, please send the error text to the control system manufacturer.

If the code for the error cause is 11, select a larger telegram as appropriate using STEP 7 HW config or

select a smaller telegram under $MN_DRIVE_TELEGRAM_TYPE.

Programm continuation: Switch control OFF - ON.

380050 PROFIBUS/PROFINET: Multiple assignment of inputs on address %1

Parameters: %1 = Logical address

Explanation: Multiple assignments of input data have been detected in the logical address space. Logical address: Base address of the address area defined several times

Reaction: Channel not ready.

NC Start disable in this channel.

Interface signals are set.

Alarm display.

Remedy: The address partitioning should be checked as follows:
Check for multiple assignments in the following machine data:
- MD13050 $MN_DRIVE_LOGIC_ADDRESS[0] - MD13050 $MN_DRIVE_LOGIC_ADDRESS[n-1] : n = highest axis index on control system
- MD12970 $MN_PLC_DIG_IN_LOGIC_ADDRESS, MD12971 $MN_PLC_DIG_IN_NUM : PLC address area for digital inputs
- MD12978 $MN_PLC_ANA_IN_LOGIC_ADDRESS, MD12979 $MN_PLC_ANA_IN_NUM : PLC address area for analog inputs
If no inconsistencies can be found in the parameters, compare these machine data with the configuration in SDB (STEP 7 project). In particular, check that the lengths configured for the individual slots do not result in area overlaps. When you find the cause of the error, change the machine data and/or SDB.

Programm continuation: Switch control OFF - ON.

380051 PROFIBUS/PROFINET: Multiple assignment of outputs on address %1

Parameters: %1 = Logical address

Explanation: Multiple assignments of input data have been detected in the logical address space. Logical address: Base address of the address area defined several times

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: The address partitioning should be checked as follows:
Check for multiple assignments in the following machine data:
- MD13050 $MN_DRIVE_LOGIC_ADDRESS[0] - MD13050 $MN_DRIVE_LOGIC_ADDRESS [n-1] : n = highest axis index on control system
- MD12974 $MN_PLC_DIG_OUT_LOGIC_ADDRESS, MD12975 $MN_PLC_DIG_OUT_NUM : PLC address area for digital outputs
- MD12982 $MN_PLC_ANA_OUT_LOGIC_ADDRESS, MD12983 $MN_PLC_ANA_OUT_NUM : PLC address area for analog outputs
If no inconsistencies can be found in the parameters, compare these machine data with the configuration in the SDB (STEP 7 project). In particular, check that the lengths configured for the individual slots do not result in area overlaps. When you find the cause of the error, change the machine data and/or SDB.

Programm continuation: Switch control OFF - ON.

380060 PROFIBUS/PROFINET: Alarm %1 on logical address %2 from unassigned slave/device

Parameters: %1 = Alarm class
%2 = Logical address

Explanation: SDB contains a slave/device which is not assigned in the NC via the MD parameters (see also alarm 380050/051). The slave/device is however connected to the PROFIBUS/PROFINET, and has reported an alarm.
Alarm class:
- 01 = Station return (or arrival)
- 02 = Station failure
Display alarm, further operation with the NC is possible.

Reaction: Alarm display.

Remedy:
- Enter machine data or
- Modify SDB or
- Disconnect the slave/device from the PROFIBUS/PROFINET or
- Acknowledge the alarm.

Programm continuation: Clear alarm with the Delete key or NC START.

380070 PROFIBUS/PROFINET: No input slot available for basic address %1 (length %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Explanation: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
Length=1 indicates a digital input.
Length=2 indicates a analog input.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTIN.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTIN.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Programm continuation: Switch control OFF - ON.

380071 PROFIBUS/PROFINET: No output slot available for basic address %1 (size %2)

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Explanation: An incorrect logical base address was specified for a digital or analog input. Either no slot has been configured for this base address or the requested area extends beyond the end of the slot.
For length =1 it is a digital output,
For length =2 it is an analog output.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Enter correct base addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Programm continuation: Switch control OFF - ON.

380072 PROFIBUS/PROFINET: Output slot for basic address %1 (size %2) not allowed

Parameters: %1 = Logical base address of the requested area
%2 = Size of the area in bytes

Explanation: An incorrect logical base address was set for a digital or analog output, the area is resides in the access range of the PLC (PIQ, base addresses < 256).
For length =1 it is a digital output,
For length =2 it is an analog output.

Reaction: Channel not ready.
NC Start disable in this channel.
Interface signals are set.
Alarm display.

Remedy: Only use addresses outside the PLC process image (e.g. >= 256) for output slots.
Enter correct basic addresses in the machine data:
- For length=1: Correct machine data MN_HW_ASSIGN_DIG_FASTOUT.
- For length=2: Correct machine data MN_HW_ASSIGN_ANA_FASTOUT.
- NCK restart
If the error cannot be eliminated by this procedure, please make a note of the error text and contact the control system manufacturer.

Programm continuation: Switch control OFF - ON.

380075 PROFIBUS/PROFINET: Failure of DP I/O bus %2 slave/device %1 logical start address %3

Parameters: %1 = Slave/device address
%2 = Number of the affected bus
%3 = PROFINET: Logical start address with input-IOPS=BAD

Explanation: Failure of a PROFIBUS/PROFINET slot used by the NCK for digital or analog I/Os.
PROFINET: Input IOPS=BAD was reported at logical start address

Reaction: Alarm display.

Remedy: Check that the slave/device is operating correctly (all slaves/devices must be included in the bus, green LEDs).

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380076 PROFIBUS/PROFINET: No DO1 message frame: Bus %2 slave/device %1

Parameters: %1 = Slave/device address
%2 = Number of the affected bus

Explanation: Note for the system setup engineer: A PROFIBUS slave/PROFINET device used as an NCK drive does not have a valid DO1 message frame assignment (see MD13120 $MN_CONTROL_UNIT_LOGIC_ADDRESS with the STEP 7 configuration).
This alarm is indicating, among other things, that the alarm time-of-day synchronization is not working between the controller and this slave/device.

Reaction: Alarm display.

Remedy: Enter a valid value in MD13120 $MN_CONTROL_UNIT_LOGIC_ADDRESS.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380077 PROFIBUS/PROFINET: Too many DOs: Currently minimum %2, maximum %3 in DO group %1

Parameters: %1 = DO group
%2 = Current number of DOs
%3 = Maximum permissible number of DOs

Explanation: Note for the system setup engineer: The number of equivalent DOs ("drive objects" group) on all busses (configured and connected) exceeds predefined limit values.
The services linked to these DOs (e.g. time synchronization, alarm display, HMI diagnostics, HMI data archiving) can no longer be guaranteed for all DOs in this group.
The following different DO groups exist (see parameter %1):
0 = Device (CU, DO1)
1 = Communications (CU-LINK)
2 = Drive (SERVO, VECTOR)
3 = Infeed (ALM etc.)
4 = Terminal block (TB)
5 = Terminal module (TM)

Reaction: Alarm display.

Remedy: Reduce the number of devices (containing DOs of this type) on the bus.
Use a more powerful type of controller (one which supports more DOs).

Programm continuation: Switch control OFF - ON.

380500 PROFIBUS/PROFINET: Fault on drive %1, code %2, value %3, time %4

Parameters: %1 = Axis
%2 = Fault code of drive (P947(/945)/P824)
%3 = Fault value of drive ((P949/P826)
%4 = Fault time of drive (P948/P825)

Explanation: Contents of fault memory of assigned drive.

Reaction: Alarm display.

Remedy: See drive documentation for fault codes/fault values.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380501 PROFIBUS/PROFINET: Fault on bus, slave/device, DO ID %1, code %2, value %3, time %4

Parameters: %1 = 8 bit bus number, 8 bit slave/device number, 16 bit DO ID
%2 = Fault code of drive (P947)
%3 = Fault value of the drive (P949)
%4 = Fault time of the drive (P948)

Explanation: Contents of the fault memory of the assigned slave/device.

Reaction: Alarm display.

Remedy: See drive documentation for fault codes/fault values.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

380502 PROFIBUS/PROFINET: Bus %1, slave/device %2 configuration changed

Parameters: %1 = Bus number
%2 = Slave/device address

Explanation: The bus configuration has changed.
Causes:
- First commissioning
- New slave/device recognized on the bus

Reaction: Interface signals are set.
Alarm display.

Remedy: In order to operate the bus with the new configuration, an additional restart will be required.

Programm continuation: Switch control OFF - ON.

380503 PROFIBUS/PROFINET: Bus %1 configuration changed

Parameters: %1 = Bus number

Explanation: A new SDB with a modified configuration has been provided.
The new settings will be activated only at the next bus power up.

Reaction: Interface signals are set.
Alarm display.

Remedy: In order to operate the bus with the new configuration, an additional restart will be required.

Programm continuation: Switch control OFF - ON.

5.5 PLC alarms

To view detailed description about individual alarms directly on the PPU, proceed as follows:

1. Press this key on the PPU to enter the alarm operating area.

2. Select the desired alarm.

3. Press this key to open the online help for the selected alarm.

Note: You can further press this softkey in the current help screen to show a complete list of all SINUMERIK 808D/SINUMERIK 808D ADVANCED alarms. In addition, you can also use the following softkey to search for a specific alarm by number in this list:

4. Pressing this softkey exits the help system.

400000 PLC STOP %1

Explanation: PLC not in cyclic mode. Travel with the machine is not possible.
%1: 1 Ready (User program has not been started)
2 Break (User program has been interrupted)
3 Error (Other PLC alarm with PLC Stop active)

Reaction: Alarm display.

Remedy: Rectify other PLC alarm;
set PLC stop from Startup menu
or test user program.

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

400001 System error %2

Explanation: %1 : Type number
With this alarm, internal alarm states are displayed that, in conjunction with the transferred error number, provide information on the cause and location of the error.

Reaction: PLC Stop

Remedy: Notify Siemens of this error together with the error message.

Programm continuation: Switch control OFF - ON.

400002 System error %1

Explanation: %1 : Type number
Internal error states are displayed with this alarm. An error number is also specified to provide further details about the cause and location of the error.

Reaction: PLC Stop

Remedy: Report this error to Siemens along with the type number.

Programm continuation: Switch control OFF - ON.

400003 System error, connection error to MCP

Explanation: This alarm reports a communication error to the MCP.

Reaction: PLC Stop

Remedy: Check MCP connection.

Programm continuation: Clear alarm with the Delete key or NC START.

400004 Code error: %2 network %1

Explanation: %1 : Network number
%2 : Internal error code, module type
The user program contains an operation which is not supported by the control.
Type 1: Incorrect operand
Type 2: Incorrect operation
Type 3: Timer already used (TON or TONR)
Type 4: User program too large - memory overflow
Type 5: Excessive number of internal jump marks
Type 6: Excessive number of global jump marks
Type 7: Excessive number of SBRs
Type 8: Link error, unresolved references

Reaction: PLC Stop

Remedy: Modify and reload user program.

Programm continuation: Switch control OFF - ON.

400005 PLC stop specified from start-up menu

Explanation: User program is not being processed

Reaction: Alarm display.

Remedy: Switch controller off/on

Programm continuation: Switch control OFF - ON.

400006 Loss of remanent PLC data

Explanation: 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.

Programm continuation: Clear alarm with the Delete key or NC START.

400007 Operand error: %3 %2 network %1

Explanation: %1 : Network number
%2 : Block type, block number.
%3 : Variable address

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.

Programm continuation: Switch control OFF - ON.

400008 Programming tool - version is not compatible %1 %2

Explanation: %1 : Programming tool version

This version is not compatible with the product version of the control system.

Reaction: PLC Stop

Remedy: Translate the user program using a suitable programming tool version and load in the control.

Programm continuation: Switch control OFF - ON.

400009 Computing time overrun at PLC level: %2 network %1

Explanation: %1 : Network number

%2 : Module type

Check user program of the corresponding network displayed.

Reaction: PLC Stop

Remedy: Change user program

Programm continuation: Switch control OFF - ON.

400010 Arithmetic error in user program: Type %2 network %1

Explanation: Check user program in the specified network.

%1 Network number, module ID

%2 = 1: Division by zero using fixed-point arithmetic

2: Floating-point arithmetic

Reaction: PLC Stop

Remedy: Change user program.

Programm continuation: Switch control OFF - ON.

400011 Maximum number of subroutine levels exceeded: %2 network %1

Explanation: %1 Network number

%2 Module ID

Check user program in the specified network.

Reaction: PLC Stop

Remedy: Change user program.

Programm continuation: Switch control OFF - ON.

400012 Error affecting indirect addressing: %2, network %1

Explanation: %1 Network number

%2 Module ID

Check user program in the specified network.

Reaction: PLC Stop

Remedy: Change user program.

Programm continuation: Switch control OFF - ON.

400013 PLC user program is incorrect

Explanation: The PLC user program in the control is incorrect or is not available.

Reaction: PLC Stop

Remedy: Reload PLC user program.

Programm continuation: Switch control OFF - ON.

400014 PROFINET - I/O ramp-up fault type: %1

Explanation: %1: 1 PROFINET - I/O not ramped-up
2 Software version NC - PLC incompatibility
3 Number of slots per function exceeded
4 PROFINET I/O server not ready

Reaction: PLC Stop

Remedy: Type 1: Check PROFINET hardware and/or MD 11240
Types 2 to 3: Report error to Siemens
Type 4: 828D - Check and/or replace PCU hardware and/or check MD 11240

Programm continuation: Switch control OFF - ON.

400015 PROFINET - IO I/O fault: log. addr. %1 bus/station: %2

Explanation: The PLC-UP is using I/O addresses which are not available.
%1 Logical I/O address
%2 Bus number/station number
Causes of error:
Bus peripheral has no voltage
Bus address set incorrectly
Bus connection faulty
Active MD 11240 (SDB configuration) is set incorrectly

Reaction: PLC Stop

Remedy: Rectify the error using the error cause

Programm continuation: Switch control OFF - ON.

400017 PLC TOOLMAN: missing table in DB9900

Explanation: The PLC TOOLMAN cannot find one of the tables 9900, 9901 or 9902.

Reaction: PLC Stop

Remedy: Create the missing table(s).

Programm continuation: Switch control OFF - ON.

400018 PLC TOOLMAN: Spindle or load location invalid

Explanation: The PLC TOOLMAN does not know the specified spindle or load location number.

Reaction: PLC Stop

Remedy: Specify the correct number.

Programm continuation: Switch control OFF - ON.

400019 PLC maintenance planner: Error in DB 9903 or 9904

Explanation: PLC maintenance planner: DBs 9903 and 9904 must be present and must have the same length.

Reaction: PLC Stop

Remedy: Create block(s) correctly.
Programm continuation: Switch control OFF - ON.

400020 PLC mainentance request %1 : Initial data not plausible

Explanation: 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 inititial data according to the documentation.
Programm continuation: Clear alarm with the Delete key or NC START.

400021 Wait until I/O access is possible

Explanation: I/O access is not possible at the current time.
Reaction: Alarm display.
Remedy:
Programm continuation: Clear alarm with the Delete key or NC START.

400022 The PLC functionality: %2 is not permitted.

Explanation: PLC functionality: %2
The specified PLC functionality is not supported.
Reaction: Alarm display.
Remedy: You are not permitted to use the specified PLC functionality.
Programm continuation: Clear alarm with the Delete key or NC START.

400023 Arithmetic problem in UP (REAL-Operation) OB/SBR/INT %1 Netzwerk %2

Explanation: Problems with REAL operands during the PLC-UP execution
%1 Module No.
%2 Network No.
Reaction: Alarm display.
Remedy: The variables used have to be checked for valid values.
Programm continuation: Clear alarm with the Delete key or NC START.

400024 Dynamically managed memory used up, area %1

Explanation: 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
Programm continuation: Clear alarm with the Delete key or NC START.

400025 PLC Ctrl Energy: Error in DB 9906

Explanation: 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.

Programm continuation: Switch control OFF - ON.

400026 Alarm PLC cycle time, cycle time %1

Explanation: PLC cycle time has exceeded the alarm threshold
%1 = xxxx: duration of the previous PLC cycle in µs

Reaction: Alarm display.

Remedy: User project is almost too large, too many operations or too many loops

Programm continuation: Clear alarm with the Delete key or NC START.

400027 Error PLC cycle time, cycle time %1

Explanation: PLC cycle time has exceeded the maximum value
%1 = xxxx: duration of the previous PLC cycle in µs

Reaction: PLC Stop

Remedy: User project too large, too many operations or too many loops

Programm continuation: Switch control OFF - ON.

400028 Option 32000 LadderSteps requires hardware PPU2xx.3 or higher

Explanation: PLC option 32000 LadderSteps cannot be implemented on this hardware
The option must be reset

Reaction: Alarm display.

Remedy: PLC option cannot be implemented in this product variant

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

400029 Code warning %2 in the network %1

Explanation: Too many branches in the network
%1 : Network number
%2 : Error code 13, block type

Reaction: Alarm display.

Remedy: Network contains too many branches; distribute logic better over a number of networks

Programm continuation: Alarm display showing cause of alarm disappears. No further operator action necessary.

400030 System error, USB machine control panel unsuitable

Explanation: Connected USB machine control panel is not suitable for this CNC.

Reaction: PLC Stop

Remedy: Replace USB-MCP

Programm continuation: Clear alarm with the Delete key or NC START.

400031 System error, the project is not compatible with NCK

Explanation: This project has not been released for this machine.

Reaction: PLC Stop
Remedy: Load original project.
Programm continuation: Clear alarm with the Delete key or NC START.

400032 System error, the project cannot run on this PLC version

Explanation: The project comprises instructions which are not known in this PLC version
Reaction: PLC Stop
Remedy: Load original project.
Programm continuation: Clear alarm with the Delete key or NC START.

400033 Permissible number of commands exceeded: %2 Network %1

Explanation: Permissible number of commands exceeded
%1 : Network number
%2 : Current number of commands, block
Reaction: PLC Stop
Remedy: The user project comprises too many commands, shorten user project or activate the option number of commands
Programm continuation: Clear alarm with the Delete key or NC START.

400034 Alarm %1 does not function. The data block DB9913 (ALARM_INI) is not present.

Explanation: Data block DB9913 together with the configuration of the alarm reaction, cancel criterion and channel assignment are not included in the user project.
%1 : Alarm number of the activated alarm
Reaction: Alarm display.
Remedy: In the PLC programming tool, the special data block DB9913 must be copied from "Libraries" into the project. Alarm reaction, cancel criterion and channel assignment must be specified accordingly. The project must be loaded into the PLC.
Programm continuation: Switch control OFF - ON.

5.6 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 machine manufacturer PLC program,** refer to relevant descriptions of machine tool for detailed information about the PLC user alarms	

System responses 6

6.1 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.

6.2 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.

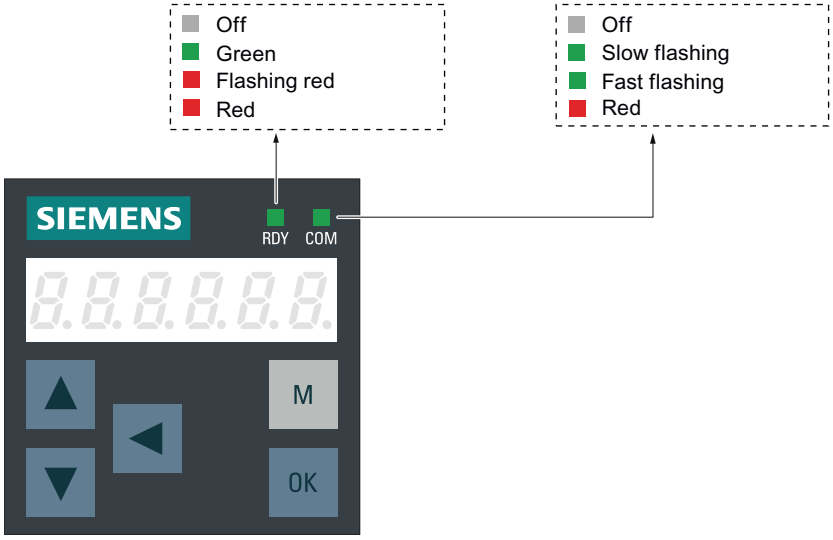
Identifier	NOCLEAR
Effects	The clear information is only required for the internal pseudo alarm number. EXBSAL_NOMOREALARMS.

SINAMICS V70 faults and alarms

7

7.1 Overview

Two LED status indicators (RDY and COM) are available to indicate drive readiness status and communication status respectively.



You can find detailed information about the status indications in the table below:

Status indicator	Color	Status	Description
RDY	-	Off	24 V control board power supply is missing
	Green	Continuously lit	The enable signal is available
	Red	Continuously lit	The enable signal is missing or the drive is in the startup state
		Flashing at 1 Hz	Alarms or faults occur
	Red and Orange	Flashing alternatively at an interval of 0.5 s	The servo drive is located
COM	-	Off	Communication with CNC is not active
	Green	Flashing at 0.5 Hz	Communication with CNC is active
		Flashing at 2 Hz	SD card operating (read or write)
	Red	Continuously lit	Communication with CNC is in error

Differences between faults and alarms

The differences between faults and alarms are as follows:

Type	Description
Faults	**What happens when a fault occurs?** • If the servo motor is running, it stops running. • If the servo motor is not running, it cannot run. **How to eliminate a fault?** • Remove the cause of the fault. • Acknowledge the fault.
Alarms	**What happens when an alarm occurs?** • The servo motor can run normally. **How to eliminate an alarm?** • The alarm acknowledges itself. If the cause of the alarm is no longer present, it automatically resets itself.

Fault reactions

The following fault reactions are defined:

Reaction	Description
NONE	No reaction when a fault occurs.
OFF1	Servo motor ramps down to stop.
OFF2	Servo motor coasts down to stop.
OFF3	Servo motor stops quickly (emergency stop).
ENCODER	Encoder fault causes OFF2.

Fault acknowledgements

The acknowledgement methods for faults are specified as follows:

Acknowledgement	Description
POWER ON	The fault is acknowledged by a POWER ON (switch servo drive off and on again). **NOTE:** If this action has not eliminated the fault cause, the fault is displayed again immediately after power-on.
IMMEDIATELY	Faults disappear immediately after the fault causes have been eliminated. **NOTE:** • These faults can also be acknowledged by a POWER ON operation. • If this action has not eliminated the fault cause, the fault will continue to be displayed after acknowledgment.
PULSE INHIBIT	The fault can only be acknowledged with a pulse inhibit. The same options are available for acknowledging as described under acknowledgment with IMMEDIATELY.

7.2 List of faults and alarms

This section lists common faults and alarms that can occur on SINAMICS V70. In case of other faults and alarms not listed below, you can call the help information on the PPU by proceeding through the following steps:

1. Select the alarm operating area.

2. Press this softkey to display all faults and alarms occured. Note that the drive faults and alarms begin with "2" instead of "F" or "A" in this window.

3. Select the desired alarm using the cursor keys.

4. Press this key to call the help information for the selected alarm.

Note: You can further press this softkey in the current help screen to show a complete list of all V70 faults and alarms. In addition, you can also use the following softkey to search for a specific fault or alarm by number in this list:

5. Pressing this softkey exits the help system.

Fault list

Fault	Cause	Remedy
F1000: Internal software error Reaction: OFF2 Acknowledgement: POWER ON	An internal software error has occurred.	• Evaluate fault buffer. • Carry out a POWER ON (power off/on) for all components. • Upgrade firmware to later version. • Contact the Hotline. • Replace the Control Unit.
F1001: Floating Point exception Reaction: OFF2 Acknowledgement: POWER ON	An exception occurred during an operation with the Floating Point data type.	• Carry out a POWER ON (power off/on) for all components. • Upgrade firmware to later version. • Contact the Hotline.
F1002: Internal software error Reaction: OFF2 Acknowledgement: IMMEDIATELY	An internal software error has occurred.	• Carry out a POWER ON (power off/on) for all components. • Upgrade firmware to the latest version. • Contact the Hotline.
F1003: Acknowledgement delay when accessing the memory Reaction: OFF2 Acknowledgement: IMMEDIATELY	A memory area was accessed that does not return a "READY".	• Carry out a POWER ON (power off/on) for all components. • Contact the Hotline.
F1015: Internal software error Reaction: OFF2 Acknowledgement: POWER ON	An internal software error has occurred.	• Carry out a POWER ON (power off/on) for all components. • Upgrade firmware to the latest version. • Contact the Hotline.
F1018: Booting has been interrupted several times Reaction: NONE Acknowledgement: POWER ON	Module booting was interrupted several times. As a consequence, the module boots with the factory setting. Possible reasons for booting being interrupted: • Power supply interrupted. • CPU crashed. • Parameterization invalid. After this fault is output, then the module is booted with the factory settings.	• Carry out a POWER ON (power off/on). After switching on, the module reboots from the valid parameterization (if available). • Restore the valid parameterization. Examples: • Carry out a first commissioning, save, carry out a POWER ON (switch-off/switch-on). • Load another valid parameter backup (e.g. from the memory card), save, carry out a POWER ON (switch-off/switch-on). Note: If the fault situation is repeated, then this fault is again output after several interrupted boots.
F1030: Sign-of-life failure for master control Reaction: OFF3 Acknowledgement: IMMEDIATELY	For active PC master control, no sign-of-life was received within the monitoring time.	Contact the Hotline.

Fault	Cause	Remedy
F1611: SI CU: Defect detected Reaction: OFF2 Acknowledgement: IMMEDIATELY	The drive-integrated "Safety Integrated" (SI) function on the Control Unit (CU) has detected an error and initiated an STO	• Carry out a POWER ON (power off/on) for all components. • Upgrade software. • Replace the Control Unit.
F1910: Drive Bus: Setpoint timeout Reaction: OFF3 Acknowledgement: IMMEDIATELY	The reception of setpoints from the Drive Bus interface has been interrupted. • Bus connection interrupted. • Controller switched off. • Controller set into the STOP state.	Restore the bus connection and set the controller to RUN.
F1911: Drive Bus clock cycle synchronous operation clock cycle failure Reaction: OFF1 Acknowledgement: IMMEDIATELY	The global control telegram to synchronize the clock cycles has failed - in cyclic operation - for several Drive Bus clock cycles or has violated the time grid specified in the parameterizing telegram over several consecutive Drive Bus clock cycles.	• Check the physical bus configuration (cable, connector, Drive Bus terminator, shielding, etc.). • Check whether communication was briefly or permanently interrupted. • Check the bus and controller for utilization level (e.g. bus cycle time was set too short).
F1912: Clock cycle synchronous operation sign-of-life failure Reaction: OFF1 Acknowledgement: IMMEDIATELY	The maximum permissible number of errors in the controller sign-of-life (clock synchronous operation) has been exceeded in cyclic operation.	• Physically check the bus (cables, connectors, terminating resistor, shielding, etc.). • Correct the interconnection of the controller sign-of-life. • Check whether the controller correctly sends the sign-of-life. • Check the permissible telegram failure rate. • Check the bus and controller for utilization level (e.g. bus cycle time was set too short).
F7011: Motor overtemperature Reaction: OFF2 Acknowledgement: IMMEDIATELY	• Motor overloaded • Motor surrounding air temperature too high • Wire breakage or sensor not connected • Motor temperature model incorrectly parameterized	• Reduce the motor load. • Check the surrounding air temperature and the motor ventilation. • Check the wiring and the connection. • Check the motor temperature model parameters.
F7085: Open-loop/closed-loop control parameters changed Reaction: NONE Acknowledgement: IMMEDIATELY	Open-loop/closed-loop control parameters have had to be changed for the following reasons: • As a result of other parameters, they have exceeded the dynamic limits. • They cannot be used due to the fact that the hardware detected not having certain features.	It is not necessary to change the parameters as they have already been correctly limited.

Fault	Cause	Remedy
F7403: Lower DC link voltage threshold reached Reaction: OFF1 Acknowledgement: IMMEDIATELY	The DC link voltage monitoring is active and the lower DC link voltage threshold was reached in the "Operation" state.	• Check the line supply voltage. • Check the infeed. • Reduce the lower DC link threshold. • Switch out (disable) the DC link voltage monitoring.
F7404: Upper DC link voltage threshold reached Reaction: OFF2 Acknowledgement: IMMEDIATELY	The DC link voltage monitoring is active and the upper DC link voltage threshold was reached in the "Operation" state.	• Check the line supply voltage. • Check the infeed module or the brake module. • Increase the upper DC link voltage threshold. • Switch out (disable) the DC link voltage monitoring.
F7410: Current controller output limited Reaction: OFF2 Acknowledgement: IMMEDIATELY	The condition "I_act = 0 and Uq_set_1 longer than 16 ms at its limit" is present and can be caused by the following: • Motor not connected or motor contactor open. • No DC link voltage present. • Motor Module defective.	• Connect the motor or check the motor contactor. • Check the DC link voltage. • Check the Motor Module.
F7411: Drive: Flux controller output limited Reaction: OFF2 Acknowledgement: IMMEDIATELY	The specified flux setpoint cannot be reached although 90% of the maximum current has been specified. • Incorrect motor data. • Motor data and motor configuration do not match. • The current limit has been set too low for the motor. • Induction motor (encoderless, open-loop controlled) in I^2t limiting.	• Correct the motor data. • Check the motor configuration. • Reduce the induction motor load.

Fault	Cause	Remedy
F7412: Commutation angle incorrect (motor model) Reaction: ENCODER Acknowledgement: IMMEDIATELY	An incorrect commutation angle was detected that can result in a positive coupling in the speed controller. Possible causes: • The motor encoder is incorrectly adjusted with respect to the magnet position. • The motor encoder is damaged. • Data to calculate the motor model has been incorrectly set. • Pole position identification might have calculated an incorrect value when activated. • The motor encoder speed signal is faulted. • The control loop is instable due to incorrect parameterization.	• If the encoder mounting was changed, re-adjust the encoder. • Replace the defective motor encoder. • Correctly set the motor stator resistance, cable resistance and motor-stator leakage inductance. Calculate the cable resistance from the cross-section and length, check the inductance and stator resistance using the motor data sheet, measure the stator resistance, e.g. using a multimeter - and if required, again identify the values using the stationary motor data identification. • With pole position identification activated, check the procedure for pole position identification and force a new pole position identification procedure by means of de-selection followed by selection.
F7414: Encoder serial number changed Reaction: ENCODER Acknowledgement: IMMEDIATELY	• The encoder was replaced. • A third-party, build-in or linear motor was re-commissioned. • The motor with integrated and adjusted encoder was replaced. • The firmware was updated to a version that checks the encoder serial number.	For the first two causes: Carry out an automatic adjustment using the pole position identification routine. Acknowledge the fault. Initiate the pole position identification routine. Then check that the pole position identification routine is correctly executed. SERVO: If a pole position identification technique is selected, and if p0301 does not contain a motor type with an encoder adjusted in the factory, then p1990 is automatically activated. Mechanically adjust the encoder. Accept the new serial number. For the last two causes: Accept the new serial number with p0440 = 1.
F7420: Drive: Current setpoint filter natural frequency > Shannon frequency Reaction: NONE Acknowledgement: IMMEDIATELY	One of the filter natural frequencies is greater than the Shannon frequency (2KHz). Filter 1 (p1658, p1660) Filter 2 (p1663, p1665) Filter 3 (p1668, p1670) Filter 4 (p1673, p1675)	• Reduce the numerator or denominator natural frequency of the current setpoint filter involved at the control system side. • Switch out the filter involved (p1656).
F7450: Standstill monitoring has responded Reaction: OFF1 Acknowledgement: IMMEDIATELY	After the standstill monitoring time expired, the drive left the standstill window. • Position loop gain too low. • Position loop gain too high (instability/ oscillation). • Mechanical overload. • Connecting cable, motor/drive converter incorrect (phase missing, interchange).	Check the causes and resolve.

Fault	Cause	Remedy
F7452: Following error too high Reaction: OFF1 Acknowledgement: IMMEDIATELY	The difference between the position setpoint position actual value (following error dynamic model) is greater than the tolerance. • The drive torque or accelerating capacity exceeded. • Position measuring system fault. • Position control sense incorrect. • Mechanical system locked. • Excessively high traversing velocity or excessively high position reference value (setpoint) differences.	Check the causes and resolve.
F7801: Motor overcurrent Reaction: OFF2 Acknowledgement: IMMEDIATELY	The permissible motor limit current was exceeded. • Effective current limit set too low. • Current controller not correctly set. • Motor was braked with an excessively high stall torque correction factor. • Up ramp was set too short or the load is too high. • Short-circuit in the motor cable or ground fault. • Motor current does not match the current of Motor Module.	• Reduce the stall torque correction factor. • Increase the up ramp or reduce the load. • Check the motor and motor cables for short-circuit and ground fault. • Check the Motor Module and motor combination.
F7802: Infeed or power unit not ready Reaction: OFF2 Acknowledgement: IMMEDIATELY	After an internal power-on command, the infeed or drive does not signal ready because of one of the following reasons: • Monitoring time is too short. • DC link voltage is not present. • Associated infeed or drive of the signaling component is defective.	• Ensure that there is a DC link voltage. Check the DC link busbar. Enable the infeed. • Replace the associated infeed or drive of the signaling component.
F7815: Power unit has been changed Reaction: NONE Acknowledgement: IMMEDIATELY	The code number of the actual power unit does not match the saved number.	Connect the original power unit and power up the Control Unit again (POWER ON).
F7860: External fault 1 Reaction: OFF2 Acknowledgement: IMMEDIATELY (POWER ON)	The signal "external fault 1" was triggered.	• Restart the drive. • Restore the drive to its factory default settings.

Fault	Cause	Remedy
F7900: Motor blocked/speed controller at its limit Reaction: OFF2 Acknowledgement: IMMEDIATELY	The servo motor has been operating at the torque limit longer than 1s and below the speed threshold of 120 rpm. This signal can also be initiated if the speed actual value is oscillating and the speed controller output repeatedly goes to its limit.	• Check whether the servo motor can rotate freely or not. • Check the torque limit. • Check the inversion of the actual value. • Check the motor encoder connection. • Check the encoder pulse number.
F7901: Motor overspeed Reaction: OFF2 Acknowledgement: IMMEDIATELY	The maximum permissible speed has been exceeded.	Check and correct the maximum speed (p1082).
F7995: Pole position identification not successful Reaction: OFF2 Acknowledgement: IMMEDIATELY	The pole position identification routine was unsuccessful.	Contact the Hotline.
F30001: Power unit: Overcurrent Reaction: OFF2 Acknowledgement: IMMEDIATELY	The power unit has detected an overcurrent condition. • Closed-loop control is incorrectly parameterized. • Motor has a short-circuit or fault to ground (frame). • Power cables are not correctly connected. • Power cables exceed the maximum permissible length. • Power unit defective. • Line phase interrupted.	• Check the motor data - if required, carry out commissioning. • Check the motor circuit configuration (star-delta) • Check the power cable connections. • Check the power cables for short-circuit or ground fault. • Check the length of the power cables. • Replace power unit. • Check the line supply phases. • Check the external braking resistor connection.
F30002: DC link voltage, overvoltage Reaction: OFF2 Acknowledgement: IMMEDIATELY	The power unit has detected overvoltage in the DC link. • Motor regenerates too much energy. • Device connection voltage too high. • Line phase interrupted.	• Increase the ramp-down time. • Activate the DC link voltage controller. • Use a braking resistor. • Increase the current limit of the infeed or use a larger module. • Check the device supply voltage. • Check the line supply phases.
F30003: DC link voltage, undervoltage Reaction: OFF2 Acknowledgement: IMMEDIATELY	The power unit has detected an undervoltage condition in the DC link. • Line supply failure • Line supply voltage below the permissible value. • Line supply infeed failed or interrupted. • Line phase interrupted.	• Check the line supply voltage. • Check the line supply infeed and observe the fault messages relating to it (if there are any). • Check the line supply phases. • Check the line supply voltage setting.

Fault	Cause	Remedy
F30004: Drive heat sink overtemperature Reaction: OFF2 Acknowledgement: IMMEDIATELY	The temperature of the power unit heat sink has exceeded the permissible limit value. • Insufficient cooling, fan failure. • Overload. • Surrounding air temperature too high. • Pulse frequency too high.	• Check whether the fan is running. • Check the fan elements. • Check whether the surrounding air temperature is in the permissible range. • Check the motor load. • Reduce the pulse frequency if this is higher than the rated pulse frequency.
F30005: Power unit: Overload I^2t Reaction: OFF2 Acknowledgement: IMMEDIATELY	The power unit was overloaded. • The permissible rated power unit current was exceeded for an inadmissibly long time. • The permissible load duty cycle was not maintained.	• Reduce the continuous load. • Adapt the load duty cycle. • Check the motor and power unit rated currents.
F30011: Line phase failure in main circuit Reaction: OFF2 Acknowledgement: IMMEDIATELY	At the power unit, the DC link voltage ripple has exceeded the permissible limit value. Possible causes: • A line phase has failed. • The 3 line phases are inadmissibly unsymmetrical. • The fuse of a phase of a main circuit has ruptured. • A motor phase has failed.	• Check the main circuit fuses. • Check whether a single-phase load is distorting the line voltages. • Check the motor feeder cables.
F30015: Phase failure motor cable Reaction: OFF2 Acknowledgement: IMMEDIATELY	A phase failure in the motor feeder cable was detected. The signal can also be output in the following case: The motor is correctly connected, however the closed-speed control is instable and therefore an oscillating torque is generated.	• Check the motor feeder cables. • Check the speed controller settings.
F30021: Ground fault Reaction: OFF2 Acknowledgement: IMMEDIATELY	Power unit has detected a ground fault. • Ground fault in the power cables. • Winding fault or ground fault at the motor.	• Check the power cable connections. • Check the motor.

Fault	Cause	Remedy
F30027: Precharging DC link time monitoring Reaction: OFF2 Acknowledgement: IMMEDIATELY	• The power unit DC link was not able to be pre-charged within the expected time. There is no line supply voltage connected. • The line contactor/line side switch has not been closed. • The line supply voltage is too low. • The pre-charging resistors are overheated as there were too many pre-charging operations per time unit • The pre-charging resistors are overheated as the DC link capacitance is too high. • The pre-charging resistors are overheated. • The pre-charging resistors are overheated as the line contactor was closed during the DC link fast discharge through the Braking Module. • The DC link has either a ground fault or a short-circuit. • The pre-charging circuit is possibly defective.	Check the line supply voltage at the input terminals.
F30036: Internal overtemperature Reaction: OFF2 Acknowledgement: IMMEDIATELY	The temperature inside the drive has exceeded the permissible temperature limit. • Insufficient cooling, fan failure. • Overload. • Surrounding air temperature too high.	• Check whether the fan is running. • Check the fan elements. • Check whether the surrounding air temperature is in the permissible range. Notice: This fault can only be acknowledged once the permissible temperature limit minus 5 K has been fallen below.
F30050: 24 V supply overvoltage Reaction: OFF2 Acknowledgement: POWER ON	The voltage monitor signals an overvoltage fault on the module.	• Check the 24 V power supply. • Replace the module if necessary.
F30071: No new actual values received from the power unit module Reaction: OFF2 Acknowledgement: IMMEDIATELY	The number of actual value telegrams from the power unit module that have failed has exceeded the permissible number.	Replace the module if necessary.

Fault	Cause	Remedy
F30074: Communication error between the Control Unit and Power Module Reaction: NONE Acknowledgement: IMMEDIATELY	Communications between the Control Unit (CU) and Power Unit (PU) via the interface no longer possible. The CU may have been withdrawn or is incorrectly inserted. Fault value (r0949, interpret hexadecimal): 0 hex: • a Control Unit with external 24 V supply was withdrawn from the Power Unit during operation. • with the Power Unit switched off, the external 24 V supply for the Control unit was interrupted for some time. 1 hex: The Control Unit was withdrawn from the Power Unit during operation, although the encoderless safe motion monitoring functions are enabled. This is not supported. After reinserting the Control Unit in operation, communications to the Power Unit no longer possible. 20A hex: The Control Unit was inserted on a Power Unit, which has another code number. 20B hex: The Control Unit was inserted on a Power Unit, which although it has the same code number, has a different serial number. 601 hex: The Control Unit was inserted on a Power Unit, whose power/performance class (chassis unit) is not supported.	Reinsert the Control Unit (CU) or the Control Unit adapter (CUAxx) onto the original Power Unit (PU) and continue operation. If required, carry out a POWER ON for the CU and/or the CUA.
F31100: Zero mark distance error Reaction: ENCODER Acknowledgement: PULSE INHIBIT	The measured zero mark distance does not correspond to the parameterized zero mark distance. For distance-coded encoders, the zero mark distance is determined from zero marks detected pairs. This means that if a zero mark is missing, depending on the pair generation, this cannot result in a fault and also has no effect in the system.	• Check that the encoder cables are routed in compliance with EMC. • Check the plug connections. • Check the encoder type (encoder with equidistant zero marks). • Replace the encoder or encoder cable.
F31110: Serial communications error Reaction: ENCODER Acknowledgement: PULSE INHIBIT	Serial communication protocol transfer error between the encoder and evaluation module.	• Check the encoder cable and shielding connection. • Replace the motor.

Fault	Cause	Remedy
F31111: Encoder 1: Absolute encoder internal error Reaction: ENCODER Acknowledgement: PULSE INHIBIT	The absolute encoder fault word supplies fault bits that have been set.	• Check the encoder cable connection, and make sure the encoder cables are routed in compliance with EMC. • Check the motor temperature. • Replace the encoder/motor.
F31112: Encoder 1: Error bit set in the serial protocol Reaction: ENCODER Acknowledgement: PULSE INHIBIT	The encoder sends a set error bit via the serial protocol.	• Check the encoder cable and shielding connection. • Replace the encoder/motor.
F31130: Zero mark and position error from the coarse synchronization Reaction: ENCODER Acknowledgement: PULSE INHIBIT	After initializing the pole position using track C/D, Hall signals or pole position identification routine, the zero mark was detected outside the permissible range. For distance-coded encoders, the test is carried out after passing 2 zero marks. Fine synchronization was not carried out.	• Check that the encoder cables are routed in compliance with EMC. • Check the plug connections. • If the Hall sensor is used as an equivalent for track C/D, check the connection. • Check the connection of track C or D. • Replace the encoder or encoder cable.
F31150: Encoder 1: Initialization error Reaction: ENCODER Acknowledgement: PULSE INHIBIT	The encoder functionality is not operating correctly.	• Check the encoder type used (incremental or absolute) and encoder cable. • If relevant, note additional fault messages that describe the fault in detail.
F52920: Encoder wire broken Reaction: OFF2 Acknowledgement: POWER ON	For a square-wave signal encoder (TTL, bipolar, double ended), the A*, B*, and R* signals are not inverted with respect to signals A, B, and R.	• Check whether the encoder is connected at the encoder port. • Check the encoder or cable for whether the encoder supplies TTL signals and the associated inverted signals.
F52980: Absolute encoder motor changed Reaction: OFF1 Acknowledgement: IMMEDIATELY	The servo motor with an absolute encoder is changed. Actual motor ID is different from commissioned motor ID.	The servo motor will be automatically configured after the acknowledgement of this fault.
F52981: Absolute encoder motor mismatched Reaction: OFF1 Acknowledgement: IMMEDIATELY	The motor connected with an absolute encoder cannot be operated. The servo drive in use does not support the Motor ID.	Use a suitable motor.
F52983: No encoder detected Reaction: OFF1 Acknowledgement: IMMEDIATELY	The servo drive in use does not support encoderless operation.	• Check the encoder cable connection between the servo drive and the servo motor. • Use a servo motor with encoder.

Fault	Cause	Remedy
F52984: Incremental encoder motor not configured Reaction: OFF1 Acknowledgement: IMMEDIATELY	• Commissioning of the servo motor has failed. • An incremental encoder motor is connected but not commissioned.	Configure the motor ID by setting the parameter p29000.
F52985: Absolute encoder motor wrong Reaction: OFF1 Acknowledgement: IMMEDIATELY	• Motor ID is downloaded wrong during manufacture. • The software of the servo drive does not support the Motor ID.	• Update the software. • Use a suitable motor.
F52987: Absolute encoder replaced Reaction: OFF1 Acknowledgement: IMMEDIATELY	Incorrect data of the absolute encoder.	Contact the Hotline.

Alarm list

A1009: Control module overtemperature	The temperature of the control module (Control Unit) has exceeded the specified limit value.	• Check the air intake for the Control Unit. • Check the Control Unit fan. Note: The alarm automatically disappears after the limit value has been undershot.
A1019: Writing to the removable data medium unsuccessful	The write access to the removable data medium was unsuccessful.	Remove and check the removable data medium. Then run the data backup again.
A1032: All parameters must be saved	The parameters of an individual drive object were saved, although there is still no backup of all drive system parameters. The saved object-specific parameters are not loaded the next time that the system powers up. For the system to successfully power up, all of the parameters must have been completely backed up.	Save all parameters.
A1045: Configuring data invalid	An error was detected when evaluating the parameter files saved in the non-volatile memory. Because of this, under certain circumstances, several of the saved parameter values were not able to be accepted.	Save the parameterization using the "SAVE" function on the BOP. This overwrites the incorrect parameter files in the non-volatile memory – and the alarm is withdrawn.
A1920: Drive Bus: Receive setpoints after To	Output data of Drive Bus master (setpoints) received at the incorrect instant in time within the Drive Bus clock cycle.	• Check bus configuration. • Check parameters for clock cycle synchronization (ensure To > Tdx). Note: • To: Time of setpoint acceptance • Tdx: Data exchange time

A1932: Drive Bus clock cycle synchronization missing for DSC	There is no clock synchronization or clock synchronous sign of life and DSC is selected. Note: DSC: Dynamic Servo Control	Set clock synchronization across the bus configuration and transfer clock synchronous sign-of-life.
A5000: Drive heat sink overtemperature	The alarm threshold for overtemperature at the inverter heat sink has been reached. If the temperature of the heat sink increases by an additional 5 K, then fault F30004 is initiated.	Check the following: • Is the surrounding air temperature within the defined limit values? • Have the load conditions and the load duty cycle been appropriately dimensioned? • Has the cooling failed?
A6310: DC link voltage exceeding the tolerance range	For AC/AC drive units, the measured DC voltage lies outside the tolerance range after pre-charging has been completed. The following applies for the tolerance range: 1.16 * 400 V < actual DC link voltage < 1.6 * 400 V. The alarm can only be eliminated when the drive is powered down.	Check the line supply voltage.
A7012: Motor temperature model 1/3 overtemperature	The motor temperature model 1/3 identified that the alarm threshold was exceeded.	• Check the motor load and reduce it if required. • Check the motor surrounding air temperature.
A7565: Encoder error in encoder interface	An encoder error was signaled for encoder via the encoder interface (G1_ZSW.15).	Acknowledge the encoder error using the encoder control word (G1_STW.15 = 1).
A7576: Encoderless operation due to a fault active	Encoderless operation is active due to a fault.	• Remove the cause of a possible encoder fault. • Carry out a POWER ON (power off/on) for all components.
A7965: Save required	The angular commutation offset was re-defined and has still not been saved. In order to permanently accept the new value, it must be saved in a non-volatile fashion.	This alarm automatically disappears after the data has been saved.
A7971: Angular commutation offset determination activated	The automatic determination of the angular commutation offset (encoder adjustment) is activated. The automatic determination is carried out at the next power-on command.	The alarm automatically disappears after determination.
A7991: Motor data identification activated	The motor data identification routine is activated. The motor data identification routine is carried out at the next power-on command.	The alarm automatically disappears after the motor data identification routine has been successfully completed. If a POWER ON or a warm restart is performed with motor data identification selected, the motor data identification request will be lost. If motor data identification is required, it will need to be selected again manually following ramp-up.
A30016: Load supply switched off	The DC link voltage is too low.	• Switch on the load supply. • Check the line supply if necessary.

A30031: Hardware current limiting in phase U	Hardware current limit for phase U responded. The pulsing in this phase is inhibited for one pulse period. • Closed-loop control is incorrectly parameterized. • Fault in the motor or in the power cables. • The power cables exceed the maximum permissible length. • Motor load too high. • Power unit defective. Note: Alarm A30031 is always output if, for a Power Module, the hardware current limiting of phase U, V or W responds.	Check the motor data. As an alternative, run a motor data identification. • Check the motor circuit configuration (star-delta). • Check the motor load. • Check the power cable connections. • Check the power cables for short-circuit or ground fault. • Check the length of the power cables.
A31117: Inversion error signals A/B/R Reaction: ENCODER Acknowledgement: PULSE INHIBIT	For a square-wave encoder (bipolar, double ended) signals A*, B* and R* are not inverted with respect to signals A, B and R.	• Check the encoder/cable. • Does the encoder supply signals and the associated inverted signals?
A31411: Encoder 1: Absolute encoder signals internal alarms	The absolute encoder fault word includes alarm bits that have been set.	• Check the encoder cable connection, and make sure the encoder cables are routed in compliance with EMC. • Check the motor temperature. • Replace the encoder/motor.
A31412: Encoder 1: Error bit set in the serial protocol	The encoder sends a set error bit via the serial protocol.	• Carry out a POWER ON (power off/on) for all components. • Check that the encoder cables are routed in compliance with EMC. • Check the plug connections. • Replace the encoder.
A52900: Failure during data copying	• Copying is halted. • The SD card was plugged out. • The drive is not in the stop state.	• Re-plug in the SD card. • Make sure the drive is in the stop state.

SINAMICS V60 faults and alarms

8

8.1 Overview

Alarm code	Description
A01	Unable to identify the power board
A02	Parameter validation error (CRC error, encoder type or parameter header invalid)
A03	Unable to write data to memory
A04	Control voltage is lower than 3.5 V.
A05	IGBT is detected overcurrent.
A06	Internal chip is detected overcurrent.
A07	Grounding short circuit occurs during drive initialization
A08	Signals from encoder phases U, V, W are detected all the same (all high or all low)
A09	TTL pulse error
A14	Software failure
A21	DC link voltage is higher than 405 V
A22	IGBT current exceeds the upper limit for 300 ms
A23	DC link voltage is lower than 200 V
A24	The difference between the position setpoint and the position actual value is greater than 1000 pulses (1/10 rotation).
A41	Actual motor speeds is higher than 2300 rpm
A42	IGBT becomes overheating
A43	Following error exceeds the limit
A44	Motor load exceeds nominal motor torque
A45	Enable signal from Terminal 65 is lost during normal drive running

8.2 Common faults and alarms

Alarm code	Background	Possible Cause	Remedy
A01		Power board is broken	Replace the drive with a new one
A02		The memory is damaged due to unexpected power-off during data saving	Restore default parameters
A03		Memory is damaged	Replace the drive with a new one
A04		The 24 V DC supply is abnormal	Check the 24 V DC supply
		The drive is defective	Replace the drive with a new one

Alarm code	Background	Possible Cause	Remedy
A05	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	• There is a short-circuit between terminals U, V, W and PE on the drive • Bad grounding • Motor insulation is broken • The drive is broken	• Check the wiring • Make a correct grounding • Replace the motor with a new one • Replace the drive with a new one
A06	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	• There is a short-circuit between terminals U, V, W and PE on the drive • Bad grounding • Motor insulation is broken • The drive is broken	• Check the wiring or whether the connection to U, V or W is broken • Make a correct grounding • Replace the motor with a new one • Replace the drive with a new one
A07	This alarm may occur when the DC link is connected This alarm may occur when the motor is running	• IGBT module is broken • There is a short-circuit between U, V or W and PE	• Replace the drive with a new one • Check the wiring
A08		• UVW signals of the encoder are bad • Bad cable • Bad cable shielding • Bad wiring of the shielded ground cable • There is a failure in the interface circuit of the encoder	• Replace the drive with a new one • Check the interface circuit of the encoder
A09		• Connection failure of encoder ABZ • Bad cable • Bad cable shielding • Bad wiring of the shielded grounded cable • There is a failure in the interface circuit of the encoder	• Check the wiring of encoder cable • Check the interface circuit of the encoder
A14		There is failure at the internal software	Reset by power-on
		A short-circuit occurs to the encoder	Check the wiring of encoder

Alarm code	Background	Possible Cause	Remedy
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
	This alarm may occur when the DC link is connected	• The mains supply voltage is too high • The waveform of mains supply voltage is abnormal	Check the power supply
	This alarm may occur when the motor is running	• Disconnection of the internal brake resistor • The internal brake resistor is broken	Replace the drive with a new one
		Brake loop has no enough space	• Lower the start-stop frequency • Minish the limit value of current • Minish load inertia • Use another drive and motor with higher power
A22		The motor is mechanically blocked	Check the mechanical load
		Overload	• Lighten the load • Use another drive and motor with bigger power
A23		• Circuit board fails • Fuse of the power is burnt out • Rectifier is broken	Replace the drive with a new one
		• Low supply voltage • Insufficient supply power capacity • Transient power failure	Check the power supply
A24	This alarm may occur when it's detected that the difference between the position setpoint and the position actual value is greater than 1000 pulses when the drive is in "S-2", "S-3", or "S-4" state	There is pulse inputs at the setpoint interface X5 when the drive is in "S-2", "S-3", or "S-4" state	When P38 = 1, check whether more than 1000 pulses are received at the setpoint interface X5 when the drive is in "S-2", "S-3", or "S-4" state
		There is shaft move when the drive is in "S-2", "S-3", or "S-4" state	When P38 = 1, check whether the motor shaft moves more than 1000 pulses when the drive is in "S-2", "S-3", or "S-4" state
A41	This alarm may occur when the 24 V DC supply is connected	Circuit board fails	Replace the drive with a new one
		The encoder fails	Replace the drive 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 drive with a new one
	This alarm may occur when the motor starts running	• Terminals U, V and W on the motor are wrongly connected • The encoder is wrongly wired	Check the wiring
A42		Surrounding air temperature is too high	Check the surrounding air temperature
		The drive is overloaded	Check the drive load
			Replace the drive with a new one

Alarm code	Background	Possible Cause	Remedy
A43	This alarm may occur when the 24 V DC supply is connected	Circuit board fails	Replace the drive with a new one
	The motor does not rotate or reversely rotates if the command pulse is input after DC-link and mains line are connected	• Terminals U, V and W on the motor is wrongly connected • 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 maximumly-permitted following error is too small	Set a wider value range for the detection of following error (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 bigger power
		Low speed	Check the maximum speed limitation (refer to parameter P26)
		Command pulse frequency is too high	• Lower the frequency • Check whether P36 has the right value
A44	This alarm may occur when the 24 V DC supply is connected	Circuit board fails	Replace the drive with a new one
	This alarm may occur when the DC link is connected	The rated torque is exceeded	• Check the load • Lower the start/Stop frequency • 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	• Modify the gain value • 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

NOTICE

Result

Alarms with alarm code < A41 result in free stop, while alarms with alarm code ≥ A41 result in emergency stop.

NOTICE
Cancelling an alarm Alarms with alarm code < A21 can be cancelled by power-on, while alarms with alarm code ≥ A21 can be cancelled by RST terminal.

8.3 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 show one of the following error codes:

Drive error list

Drive display	Description	Cause	Remedy
	Error 1	RAM damaged	Replace the drive
	Error 2	Flash damaged	Replace the drive
	Error 3	Program copy error	Replace the drive

8.4 Other faults

1. 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.
2. Axis position incorrect or axis does not move
   - Description: When status display on the SINAMICS V60 drive is "S-Run", the axis position is incorrect or the axis does not move.
   - Cause: Pin +PLUS or -PLUS is not well connected.
   - Remedy: Check the cable connection on pin +PLUS or -PLUS.

3. Axis does not move
   - Description: When status display on the SINAMICS V60 drive is "S-4", the axis does not move even the CNC controller has sent out pulse signals.
   - Cause: Pin +ENA or -ENA is not well connected.
   - Remedy: Check the cable connection on pin +ENA or -ENA.
4. Axis keeps running in a single direction
   - Description: When status display on the SINAMICS V60 drive is "S-Run", the axis corresponding keeps running in a single direction no matter positive signals or negative signals are given.
   - Cause: Pin +DIR or -DIR is not well connected.
   - Remedy: Check the cable connection on pin +DIR or -DIR.

Data backup

9

9.1 Internal data backup

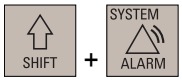
You can save the data of the limited-buffered memory via a backup copy to the permanent memory of the control system.

Note

After changing important data, it is recommended to carry out an internal data backup **immediately**.

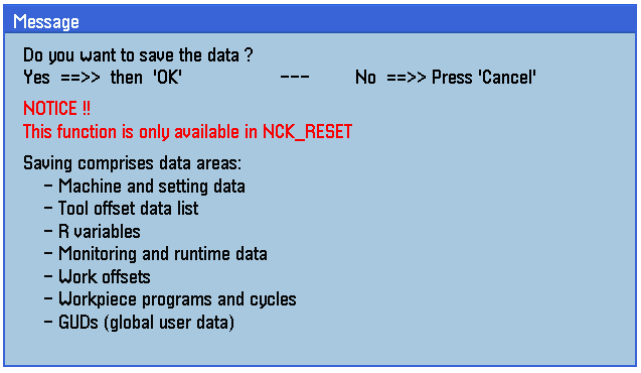
Backing up data internally

Proceed as follows to back up data internally:



1. Select the system data operating area.

2. Press this softkey, and the following message prompts:



3. Press this softkey to confirm. The control system begins to save data. Wait until the progress dialog disappears and the data saving finishes.

Note

While performing an internal data backup, you must neither operate nor turn off the control system.

Loading internally backed up data

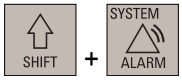
There are two methods to load the internally backed up data.

Method 1:

1. Press this key while the control system is booting.
2. Select "Reload saved user data" in the setup menu.

3. Press this key to confirm.

Method 2:

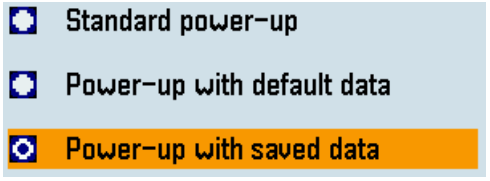


1. Select the system data operating area.

2. Open the window for selecting the startup modes.

3. Press this softkey.

4. Use the cursor keys to select the third startup mode as follows:



5. Press this softkey to confirm. The control system automatically re-starts with the saved data.

Note

The following message is displayed on the screen after the control system starts up successfully with the saved data:

You must enter the password again after you have powered up the control system with the saved data.

9.2 External data backup

You can perform external data backup in the following ways:

- Creating a data archive (Page 587)
- Copying the system data to the selected storage medium (Page 589)
- Backing up data through the Ethernet interface (Page 591)

Note

Archiving/data backup

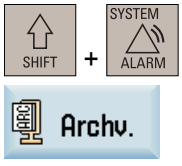
It is recommended that you regularly back up the internal SINUMERIK memory on a USB stick. You can transfer the backed up data to the SINUMERIK later on. In this way you can restore the previous status of the unit.

You must perform an external data backup if you have made major data changes or after you have commissioned the control system.

9.2.1 External data backup in a data archive

You can perform a complete data backup of the control system by creating a startup archive.

Backing up data externally in a data archive



1. Select the system data operating area.

2. Press this softkey to open the window for creating or restoring a start-up archive.

   There are three options for creating a data archive. Note that options ① and ③ are visible only with the manufacturer password.

   ①: Creates a data archive for series machine commissioning

   ②: Creates a data archive for a complete system backup

   ③: Backs up the complete system data directly on the system CompactFlash Card (CF card)

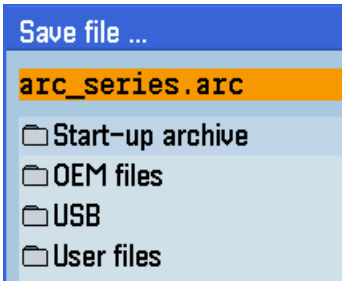
   The option ③ backs up exactly the same data with the option ②. In case of service (e.g. hardware replacement), it can be more convenient to restore the control system directly from the system CF card than from other paths. After creating a startup archive with option ③, the following option appears on the screen:

3. Select an option to create your desired data archive for backup and press this softkey to confirm.

   If you choose to create an archive with option ① or ②, you must specify a backup path for the data archive.

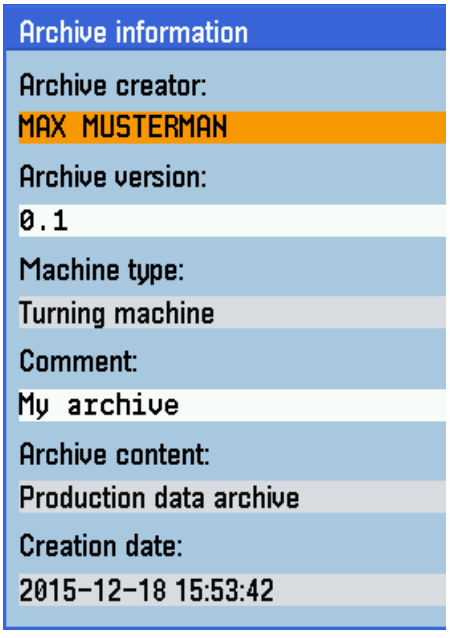
   If you choose to create an archive with option ③, it is not necessary to specify a backup path since the archive is automatically saved on the system CF card.

   The following takes option ① as an example, and the name of the data archive is "arc_series.arc" by default. You can use your favorite name for it.



4. Select your desired folder and press this key to open it.

5. Press this softkey to confirm and the archive information dialog opens.



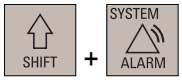
6. Specify the properties of the archive and press this softkey. The control system starts creation of the startup archive.

Note

Do not remove the USB stick in the process of data backup if you choose USB as the target directory.

Pressing <CTRL + S> when you are in any operating area (on PPU16x.3 only) creates a startup archive on the connected USB stick. In addition, it automatically saves the action log to the USB stick.

Restoring a startup archive



1. Select the system data operating area.

2. Press this softkey to open the startup archive window.

3. Select a desired option to restore the startup archive and press this softkey.

   ① Restore start-up archive
   ② Restore start-up archive from default directory

   Note that the option ② appears only after you have created an archive in the default directory.

   If you use the option ①, you also need to select the backup path and find the backup archive.

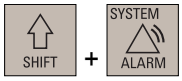
4. Press this softkey to confirm the archive information.

5. Press this softkey to continue and start restoring the startup archive. The control system restarts to complete restoring the archive.

9.2.2 External data backup of system data

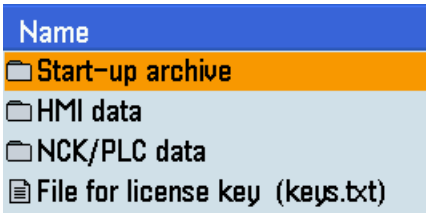
You can perform data backup of certain system data in the form of separate files.

Backing up system data in separate files



1. Select the system data operating area.

2. Press this horizontal softkey to open the system data window. Three folders and one file are available in this window.



Note that the last three entries are visible with the manufacturer password only.

3. Select a desired folder and press this key to open it.

4. Select the file that you desire to back up, and press this softkey. To copy all files in the folder, press the following softkey before you press this softkey:

5. Select a data backup path by pressing one of the available softkeys:

Backs up the files in the folder for storing the user cycles on the control system. This folder is visible with the manufacturer password.

Backs up the files onto an USB stick

Backs up the files to an external PC through the Ethernet interface. For more information, see section "External data backup through the Ethernet interface (Page 591)'.

Backs up the files in the folder for storing the manufacturer files on the control system. This folder is visible with the manufacturer password.

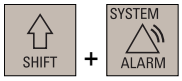
Backs up the files in the folder for storing end user files on the control system

6. Press this softkey to paste the copied data into the selected directory.

Note

Do not remove the USB stick during the backup process if you select USB as the backup path.

Loading system data in separate files (manufacturer or Siemens access level)



1. Select the system data operating area.

2. Press this softkey to open the system data window.

3. Press a softkey according to the backup path of the file.

4. Find the backed up file and press this softkey.

5. Press this softkey to open the 808D data window.

6. Paste the copies file to the corresponding folder with this softkey.

9.2.3 External data backup through the Ethernet interface

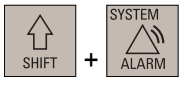
You can back up the system data files to your computer by means of a connected network drive. The network drive functions based on the Ethernet connection between the control system and a computer.

Creating and connecting a network drive

Prerequisite: A network connection has been established between the control system and the computer. For more information about establishing Ethernet connections, see section "Establishing an Ethernet connection (Page 45)".

Proceed as follows to create and connect a network drive:

1. Share a directory on your local disk on your computer.



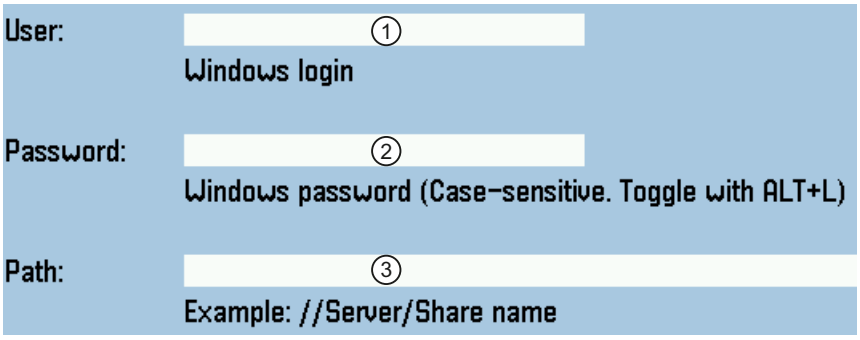
2. Select the system data operating area on the PPU.

3. Press this softkey to go to the network drive directory.

4. Press this softkey to go to the window for configuring the network drives.

5. Press this key to select a drive identifier: N1, N2, or N3.

6. Move the cursor to the following input fields:



①: Enter the user name of your Windows account

②: Enter the logon password (case sensitive) of your Windows account

③: Enter the IP address of the server and the share name of the shared directory on your computer. Example: //140.231.196.90/808D

7. Press this softkey to confirm and the configured network drive appears on the screen as follows. The drive icon is yellow if the network drive is connected successfully; otherwise, the icon is gray.

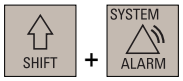
You can delete a selected network drive using this softkey.

Note

After you properly configure all the settings for the direct connection between the control system and the network drive, if the network drive connection is still invalid, contact your Windows system administrator for possible problems with your operating system configuration.

Backing up data to the network drive

Proceed as follows to perform data backup through a network drive:



1. Select the system data operating area on the PPU.

2. Press this horizontal softkey to open the system data window.

3. Navigate to the NC file that you desire to back up, and copy it to the buffer memory on the control system with this softkey.

4. Press this softkey to view the network drive(s) created.

5. Enter the desired network drive with this key.

6. Press this softkey to paste the copied file into the current directory. After the file is successfully pasted, you can find it in the corresponding directory on your computer.

Updating software

10

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/drive) before you start the update.

For information about the data backup, see section "Data backup (Page 585)".

Operating sequence

To update the control system using a USB stick, proceed as follows:

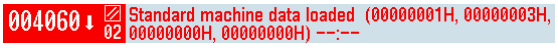
1. Before updating, copy the update file to your USB stick.
2. Switch off the control system.
3. Plug the USB stick into the USB interface in the front of the control system.
4. Switch on the control system.

5. While the control system boots, once the message "Press the SELECT key to enter set-up menu" appears, press this key.

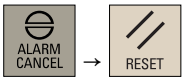
6. Use the cursor key to select the option of "Software update" in the Set-up menu, and press this key.

   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.

7. Select the update file and press this key to confirm.
8. The update process begins when the message "SW-Update initiated ..." appears.
9. Wait until the message "Restoring complete. Switch off and remove data medium!" appears.
10. Switch off the control system and remove the USB stick.
11. Switch on the control system.
12. The update process has been completed when these two error messages appear.



13. Press this key or the key combination below 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
LAD	Ladder Diagram	Ladder diagram

Abbreviation	Source of abbreviation	Meaning
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
WAB		Smooth approach and retraction
Work		Workpiece coordinate system
T	Tool	Tool
TLC	Tool Length Compensation	Tool length compensation

Abbreviation	Source of abbreviation	Meaning
TC	Tool Change	Tool change
TM	Tool Management	Tool management