MITSUBISHI ELECTRIC

Changes for the Better

Numerical Control (CNC)

Alarm Manual

M800V/M80V Series

IB-1501808(ENG)-C

Introduction

This manual describes the alarms of Mitsubishi Electric CNC.
Improper handling can cause unexpected malfunctions. To use this device correctly, be sure to read this manual before use.

Supported models of this manual are as follows:

Supported models	Abbreviations in this manual
M800VW Series	M850VW, M830VW
M800VS Series	M850VS, M830VS
M80VW Series	M80VW
M80V Series	M80V TypeA, M80V TypeB

Abbreviations for model names used in this manual are as follows:

Abbreviations	Supported models
M800V, M800V Series	M800VW Series/M800VS Series
M80V, M80V Series	M80VW Series/M80V Series
M800V/M80V, M800V/M80V Series	M800VW Series/M800VS Series/M80VW Series/M80V Series
M8V, M8V Series	

To safely use this CNC unit, thoroughly study the "Precautions for Safety" before use.
Be sure to keep this manual on hand so that users can refer to it at any time.
Also refer to the manuals on "Manual List" as necessary.

Notes on reading this manual

(1) For information about each machine tool, refer to manuals issued by the machine tool builder.
If the descriptions relating to "restrictions" and "allowable conditions" conflict between this manual and the machine tool builder's manual, the later has priority over the former.
(2) This manual is intended to contain as much descriptions as possible even about special operations.
The operations to which no reference is made in this manual should be considered "impossible".

Details described in this manual

- In this manual, the following abbreviations might be used.
  L system: Lathe system
  M system: Machining center system
  MTB: Machine tool builder
- In this manual, some "auxiliary axis" indicates PLC axis where axis indexing is performed by index position command.

Manual List

Manuals related to M800V/M80V Series are listed as follows.
These manuals are written on the assumption that all optional functions are added to the targeted model.
Some functions or screens may not be available depending on the machine or specifications set by MTB. (Confirm the specifications before use.)
The manuals issued by MTB take precedence over these manuals.

Manual	IB No.	Purpose and Contents
M800V/M80V Series Instruction Manual (1/2)	IB-1501806	♦ Explanation for display unit and screen operation ♦ Settings before machining, setup, etc.
M800V/M80V Series Instruction Manual (2/2)	IB-1501807	♦ Explanation for settings at machining, operation status, etc. ♦ Explanation for other functions
M800V/M80V Series Programming Manual (Lathe System) (1/2)	IB-1501619	♦ G code programming for lathe system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Lathe System) (2/2)	IB-1501620	♦ G code programming for lathe system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Programming Manual (Machining Center System) (1/2)	IB-1501621	♦ G code programming for machining center system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Machining Center System) (2/2)	IB-1501622	♦ G code programming for machining center system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Alarm Manual	IB-1501808	♦ Alarms
M800V/M80V Series Parameter Manual	IB-1501809	♦ Parameters

Manuals for MTBs (NC)

Manual	IB No.	Purpose and Contents
M800V/M80V Series Specifications Manual (Function)	IB-1501610	• Model selection • Outline of various functions
M800V/M80V Series Specifications Manual (Hardware)	IB-1501611	• Model selection • Specifications of hardware unit
M800VW/M80VW Series Connection and Setup Manual	IB-1501612	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800VS/M80V Series Connection and Setup Manual	IB-1501613	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800V/M80V Series PLC Development Manual	IB-1501614	• Electrical design • I/O relation (assignment, setting, connection), field network • Development environment (PLC on-board, peripheral development environment), etc.
M800V/M80V Series PLC Programming Manual (1/2)	IB-1501667	• Electrical design • Sequence programming • Explanation for instructions, functions, and parameters
M800V/M80V Series PLC Programming Manual (2/2)	IB-1501668	• Electrical design • Sequence programming • Usage examples of instructions
M800V/M80V Series PLC Interface Manual (1/2)	IB-1501804	• Electrical design • List of interface signals between NC and PLC
M800V/M80V Series PLC Interface Manual (2/2)	IB-1501805	• Electrical design • Details of interface signals between NC and PLC
M800V/M80V Series Maintenance Manual	IB-1501617	• Cleaning and replacement for each unit • Other items related to maintenance
NC Designer2 Instruction Manual	IB-1501250	• Explanation for handling NC NC Designer2
High Speed Processing Unit User's Manual	IB-1501714	• Specifications for high speed processing unit (HPU)
NC Machining AI Diagnosis NC MachiningAID Instruction Manual	IB-1501762	• Explanation for handling NC MachiningAID

Manuals for MTBs (drive section)

Manual	IB No.	Contents
MDS-E/EH Series Specifications Manual	IB-1501226	• Specifications for power supply regeneration type
MDS-E/EH Series Instruction Manual	IB-1501229	• Instruction for power supply regeneration type
MDS-EJ/EJH Series Specifications Manual	IB-1501232	• Specifications for regenerative resistor type
MDS-EJ/EJH Series Instruction Manual	IB-1501235	• Instruction for regenerative resistor type
MDS-EM/EMH Series Specifications Manual	IB-1501238	• Specifications for multi-hybrid, power supply regeneration type
MDS-EM/EMH Series Instruction Manual	IB-1501241	• Instruction for multi-hybrid, power supply regeneration type
DATA BOOK	IB-1501252	• Specifications of servo drive unit, spindle drive unit, motor, etc.
MDS-EX-CVP Series Specifications and Instruction Manual	IB-1501587	• Specifications and instruction for the power supply unit with large capacity

Manuals for MTBs (Others)

Manual	No.	Purpose and Contents
GOT2000 Series User's Manual (Hardware)	SH-081194ENG	• Outline of hardware such as part names, external dimensions, installation, wiring, maintenance, etc. of GOTs
GOT2000 Series User's Manual (Utility)	SH-081195ENG	• Outline of utilities such as screen display setting, operation method, etc. of GOTs
GOT2000 Series User's Manual (Monitor)	SH-081196ENG	• Outline of each monitor function of GOTs
GOT2000 Series Connection Manual (Mitsubishi Electric Products)	SH-081197ENG	• Outline of connection types and connection method between GOT and Mitsubishi Electric connection devices
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 1)	SH-081198ENG	• Explanation for connection types and connection method between GOT and other company's devices
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 2)	SH-081199ENG	
GOT2000 Series Connection Manual (Microcomputers, MODBUS/Fieldbus Products, Peripherals)	SH-081200ENG	• Explanation for connection types and connection method between GOT and microcomputers, MODBUS/fieldbus products, peripherals
GT SoftGOT2000 Version1 Operating Manual	SH-081201ENG	• Explanation for system configuration, screen configuration and operation method of monitoring software GT SoftGOT2000
GT Designer3 (GOT2000) Screen Design Manual	SH-081220ENG	• Outline of screen design method using screen creation software GT Designer3
GOT2000/GOT1000 Series CC-Link Communication Unit User's Manual	IB-0800351	• Explanation for handling CC-Link communication unit (for GOT2000 series/GOT1000 series)
GX Developer Version 8 Operating Manual (Startup)	SH-080372E	• Explanation for system configuration, installation, etc. of PLC development tool GX Developer
GX Developer Version 8 Operating Manual	SH-080373E	• Explanation for operations using PLC development tool GX Developer
GX Converter Version 1 Operating Manual	IB-0800004	• Explanation for operations using data conversion tool GX Converter
GX Works2 Installation Instructions	BCN-P5999-0944	• Explanation for the operating environment and installation method of GX Works2
GX Works2 Version 1 Operating Manual (Common)	SH-080779ENG	• Explanation for the system configuration of GX Works2 and the functions common to Simple project and Structured project such as parameter setting, operation method for the online function
GX Works2 Version 1 Operating Manual (Simple Project)	SH-080780ENG	• Explanation for methods for such as creating and monitoring programs in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Simple Project, Function Block)	SH-080984ENG	• Explanation for methods for such as creating function blocks, pasting function blocks to sequence programs, and operating FB library in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Structured Project)	SH-080781ENG	• Explanation for methods for such as creating and monitoring programs in Structured project of GX Works2
GX Works3 Installation Instructions	BCN-P5999-0391	• Explanation for the operating environment and installation method of GX Works3
MELSEC-Q/L/F Structured Programming Manual (Fundamentals)	SH-080782ENG	• Explanation for programming methods, types of programming languages, etc. required to create structured programs
MELSEC-Q/L Structured Programming Manual (Application Functions)	SH-080784ENG	• Explanation for specifications and functions related to application functions which can be used in structured programs
MELSEC-Q CC-Link System Master/ Local Module User's Manual	SH-080394E	• Explanation for system configuration, installation, wiring, etc. of master/local modules for CC-Link system
MELSEC iQ-R Programming Manual (CPU Module Instructions, Standard Functions/Function Blocks)	SH-081266ENG	• Explanation for instructions, general-purpose functions, and general-purpose function blocks required for programming the sequencer MELSEC iQ-R series

Reference Manual for MTBs

Manual	No.	Purpose and Contents
M800/M80/M800V/M80V Series Smart safety observation Specification manual	BNP-C3072-022	♦Explanation for smart safety observation function
M800V/M80V Series PROFIBUS-DP Specification manual	BNP-C3095-118	♦Explanation for PROFIBUS-DP communication function (English translation pending)
M800V/M80V Series EtherNet/IP Specification manual	BNP-C3095-263	♦Explanation for EtherNet/IP
M800/M80 Series FL-net Specifications manual	BNP-C3072-368	♦Explanation for FL-net
M800V/M80V Series CC-Link (Master/Local) Specification manual	BNP-C3095-089	♦Explanation for CC-Link (English translation pending)
M800/M80 Series CC-Link IE Field (Master/local) Specification manual	BNP-C3072-283	♦Explanation for CC-Link IE Field
M800/M80 Series CC-Link IE Field Basic Specification manual	BNP-C3072-337	♦Explanation for CC-Link IE Field Basic
M800V/M80V Series CC-Link IE TSN (Remote Station) Specifications manual	BNP-C3095-399	♦Explanation for CC-Link IE TSN (Remote Station)
IoT Unit Edge PC App Specification manual	BNP-C3087-008	♦Explanation for Edge PC App (English translation pending)

Precautions for Safety

Always read the specifications issued by the machine tool builder, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
Understand this numerical controller, safety items and cautions before using the unit.
This manual ranks the safety precautions into "DANGER", "WARNING" and "CAUTION".

⚠ DANGER
When the user may be subject to imminent fatalities or major injuries if handling is mistaken.

⚠ WARNING
When the user may be subject to fatalities or major injuries if handling is mistaken.

⚠ CAUTION
When the user may be subject to medium or minor injuries or when only property damage may occur, if handling is mistaken.

Note that even items ranked as " ⚠ CAUTION" may lead to serious consequences depending on the situation. All the items are important and must always be observed.
The following signs indicate prohibition and compulsory.

	This sign indicates prohibited behavior (must not do). For example,  indicates "Keep fire away".
	This sign indicated a thing that is pompously (must do). For example,  indicates "it must be grounded".

The meaning of each pictorial sign is as follows.

 CAUTION	 CAUTION rotated object	 CAUTION HOT	 Danger Electric shock risk	 Danger explosive
 Prohibited	 Disassembly is prohibited	 KEEP FIRE AWAY	 General instruction	 Earth ground

For Safe Use

Mitsubishi Electric CNC is designed and manufactured solely for applications to machine tools to be used for industrial purposes.
Do not use this product in any applications other than those specified above, especially those which are substantially influential on the public interest or which are expected to have significant influence on human lives or properties.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

Not applicable in this manual.

⚠ CAUTION

(1) Product and manual

⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.

⚠ Items not described in this manual must be interpreted as "not possible".

⚠ This manual is written on the assumption that all the applicable functions are included. Some of them, however, may not be available for your NC system. Refer to the specifications issued by the machine tool builder before use.

⚠ Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.

⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

⚠ To protect the availability, integrity and confidentiality of the NC system against cyber-attacks including unauthorized access, denial-of-service (DoS) (*1) attack, and computer virus from external sources via a network, take security measures such as firewall, VPN, and anti-virus software.
(*1) Denial-of-service (DoS) refers to a type of cyber-attack that disrupts services by overloading the system or by exploiting a vulnerability of the system.

⚠ Mitsubishi Electric assumes no responsibility for any problems caused to the NC system by any type of cyber-attacks including DoS attack, unauthorized access and computer virus.

(2) Adjustments

⚠ Do not adjust or change the parameter settings greatly as operation could become unstable.

⚠ In the explanation on bits, set all bits not used, including blank bits, to "0".

(3) Troubleshooting

❗ If the battery low warning is issued in the controller side, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may have been destroyed. Replace the battery and then reload the data.

⚠ If the battery low warning is issued in the drive unit side, immediately replace the battery. Replace the batteries while applying the drive unit's control power.

(4) Maintenance, inspection and part replacement

⚠ Do not short-circuit, charge, heat, incinerate or disassemble the battery.

⚠ Dispose of the spent battery according to local laws.

Disposal



(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.
This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.
If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:
Hg: mercury (0.0005%), Cd: cadmium (0.002%), Pb: lead (0.004%)
In the European Union there are separate collection systems for used batteries and accumulators.
Please, dispose of batteries and accumulators correctly at your local community waste collection/recycling centre.

Please, help us to conserve the environment we live in!

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSEC iQ-R, MELSOFT, GOT, CC-Link, CC-Link/LT, CC-Link IE, CC-Link IE/field, CC-Link IE/field Basic, CC-Link IE TSN, EcoMonitorLight and SLMP are either trademarks or registered trademarks of Mitsubishi Electric Corporation in Japan and/or other countries.

Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.
Microsoft®, Windows®, SQL Server®, Access® and Microsoft Edge are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
SD logo and SDHC logo are either registered trademarks or trademarks of SD-3C, LLC.
UNIX is a registered trademark of The Open Group in the United States and/or other countries.
Intel® and Pentium® are either trademarks or registered trademarks of Intel Corporation in the United States and/or other countries.
MODBUS® is either a trademark or a registered trademark of Schneider Electric USA, Inc. or the affiliated companies in Japan and/or other countries.
EtherNet/IP is a trademark of Open DeviceNet Vendor Association,Inc.
PROFIBUS-DP is either a trademark or a registered trademark of PROFIBUS User Organization.
Oracle® and JavaScript are either trademarks or registered trademarks of Oracle Corporation, the subsidiaries, or the affiliated companies in the United States and/or other countries.
VNC is a registered trademark of RealVNC Ltd. in the United States and other countries.
QR Code is a registered trademark of DENSO WAVE INCORPORATED.
TRELLIX is a registered trademark or a trademark of Musarubra US LLC or its subsidiaries in the United States and other countries.
Punchtap is licensed by EMUGE.
The term HDMI and the HDMI Logo are trademarks or registered trademarks of HDMI Licensing Administrator, Inc. in the United States and other countries.
Wi-Fi is either a trademark or a registered trademark of Wi-Fi Alliance.
BiSS is a registered trademark of iC-Haus GmbH.
SPRINT™ is a trademark of Renishaw plc.
Android and Google Chrome are either trademarks or registered trademarks of Google Inc.
iPadOS, macOS and Safari are trademarks of Apple Inc., registered in the U.S. and other countries and regions.

Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

本製品の取扱いについて

(日本語 /Japanese)
本製品は工業用 (クラス A) 電磁環境適合機器です。販売者あるいは使用者はこの点に注意し、住商業環境以外での使用をお願いいたします。

Handling of our product

(English)
This is a class A product. In a domestic environment this product may cause radio interference in which case the user may be required to take adequate measures.

본 제품의 취급에 대해서

(한국어 /Korean)
이 기기는 업무용 (A 급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에 서 사용하는 것을 목적으로 합니다 .

Contents

1 Operation Errors (M) 1

2 Stop Codes (T) 49

3 Servo/Spindle Alarms (S) 59
3.1 Servo Errors (S01/S03/S04) 60
3.2 Initial Parameter Errors (S02) 70
3.3 Safety Function Errors (S05) 72
3.4 Parameter Errors (S51) 73
3.5 Servo Warnings (S52) 74
3.6 Safety Function Warnings (S53) 77

4 MCP Alarms (Y) 79

5 System Alarms (Z) 95

6 Absolute Position Detection System Alarms (Z7*) 117

7 Distance-coded Reference Scale Errors (Z8*) 121

8 Emergency Stop Alarms (EMG) 123

9 Computer Link Errors (L) 127

10 User PLC Alarms (U) 129

11 Network Service Errors (N) 141

12 Program Errors (P) 145

13 Smart Safety Observation Alarm (V) 189
13.1 Smart Safety Observation Error (V01/V02/V03/V04/V05/V06/V07) 190
13.2 Smart Safety Observation Warning (V50/V51/V52/V53/V54) 201

1

Operation Errors (M)

M01	Dog overrun	0001

Details

[Dog overrun]

When returning to the reference position, the near-point detection limit switch did not stop over the dog, but overran the dog.

[Aux ax dog overrun]

When executing dog-type reference position, the zero point return speed is too fast or the dog length is too short.

Remedy

[Dog overrun]

- Increase the length of the near-point dog.
- Reduce the reference position return speed.

[Aux ax dog overrun]

- Lower the zero point return speed or increase the dog length.

M01	Some ax does not pass Z phase	0002

Details

One of the axes did not pass the Z-phase during the initial reference position return after the power was turned ON.

Remedy

- Move the encoder one rotation or more in the opposite direction of the reference position, and repeat reference position return.

M01	R-pnt direction illegal	0003

Details

[R-pnt direction illegal]

When manually returning to the reference position, the return direction differs from the axis movement direction selected with the AXIS SELECTION key.

[Aux ax R-pnt direction illegal]

When executing reference position return, the axis was moved in the opposite of the designated direction.

Remedy

[R-pnt direction illegal]

- The selection of the AXIS SELECTION key's +/- direction is incorrect. The error is canceled by feeding the axis in the correct direction.

[Aux ax R-pnt direction illegal]

- Move the axis in the correct direction.

M01	External interlock axis exists	0004

Details

[External interlock axis exists]

The external interlock function has activated (the input signal is "OFF") and one of the axes has entered the interlock state.

[Aux ax external interlock]

The axis interlock function is valid.

Remedy

[External interlock axis exists]

- As the interlock function has activated, release it before resuming operation.
- Check the sequence on the machine side.
- Check for any broken wires in the "interlock" signal line.

[Aux ax external interlock]

- Cancel the interlock signal.

M01	Internal interlock axis exists	0005

Details

[Internal interlock axis exists]

The internal interlock state has been entered.
The axis with absolute encoder has been detached.
A command for the manual/automatic simultaneous valid axis was issued from the automatic mode.
The manual speed command was issued while the "tool length measurement 1" signal is ON.
A travel command has been issued to an inclined axis whose basic axis is in control axis synchronization between part systems.
Selected an axis other than the 1st axis when the manual speed command was issued.
A travel command has been issued to an axis stopped by the collision detection function.
Positioning or interpolation command was issued during the spindle-mode rotary axis control mode.

[Aux ax internal interlock]

An interlock was established by the servo OFF function.

Remedy

[Internal interlock axis exists]

- The servo OFF function is valid, so release it first.
- An axis that can be removed has been issued, so perform the correct operations.
- The command is issued in the same direction as the direction where manual skip turned ON, so perform the correct operations.
- During the manual/automatic simultaneous mode, the axis commanded in the automatic mode became the manual operation axis. Turn OFF the "manual/automatic valid" signal for the commanded axis.
- Turn ON the power again, and perform absolute position initialization.
- Turn OFF the "tool length measurement 1" signal to start the program by the manual speed command.
- Cancel the control axis synchronization between part systems, then issue a travel command to the inclined axis.
- Select the 1st axis of each part system when issuing the manual speed command.
- Cancel the collision detection alarm.
- Check the program after cancelling the error by reset.

[Aux ax internal interlock]

- Cancel the servo OFF.

M01	H/W stroke end axis exists	0006

Details

The stroke end function has activated (the input signal is "OFF") and one of the axes is in the stroke end status.

Remedy

- Move the machine manually.
- Check for any broken wires in the "stroke end" signal line.
- Check for any limit switch failure.

M01	S/W stroke end axis exists	0007

Details

[S/W stroke end axis exists]

The stored stroke limit I, II, IIB or IB function has activated.

[Aux ax stored stroke limit]

The stored stroke limit was reached.

Remedy

[S/W stroke end axis exists]

- Move the machine manually.
- Correct any setting error of the parameters for the stored stroke limit.

[Aux ax stored stroke limit]

- Check the stored stroke limit setting and machine position.

M01	Chuck/tailstock stroke end ax	0008

Details

The chuck/tail-stock barrier function turned ON, and an axis entered the stroke end state.

Remedy

- Reset the alarm with reset, and move the machine in the reverse direction.

M01	Ref point return No. invalid	0009

Details

2nd reference position return was performed before 1st reference position return has been completed.

Remedy

- Execute 1st reference position return.

M01	Illegal op in mid pt sg block	0013

Details

The operation mode was changed to MDI mode or reference position return mode during single block stop at the middle point of G28/G29/G30.

Remedy

- Change the operation mode.
- Reset to clear the alarm.

M01	TLM dir. determine impossible	0017

Details

The direction for Tool length measurement cannot be determined.

Remedy

- Turn OFF TLM signal, and move the tool away from the sensor.
- Maintain a clearance of "#2709 TLM_approach" or more between the tool nose and the sensor.
- Turn ON TLM signal, and execute tool length measurement again.

M01	Interlocked during MTLM	0018

Details

If the "Tool Length Measurement 2" signal (TLMS) is set to the ON state while the axis/axes are moving, all axes are interlocked and then the mode is changed.
The interlock is automatically canceled after the mode is changed.

Remedy

- Be sure to change the mode after all axes have stopped.

M01	Sensor signal illegal ON	0019

Details

The sensor signal was already ON when the tool measurement mode (TLM) signal was validated.
The sensor signal turned ON when there was no axis movement after the tool measurement mode (TLM) signal was validated.
The sensor signal turned ON at a position within 100 μm from the final entry start position.

Remedy

- Turn the tool measurement mode signal input OFF, and move the axis in a safe direction.
- Disabling the sensor signal also clears the operation alarm.

(Note) When the "tool length measurement 1" signal is disabled, the axis can be moved in either direction. Pay attention to the movement direction.

M01	Ref point retract invalid	0020

Details

Reference position retract was performed while the coordinates had not been established.

Remedy

- Execute reference position return.

M01	Tool ofs invld after R-pnt	0021

Details

Reference position return had been performed during the tool retract and return, which invalidated the tool compensation amount after the reference position return.

Remedy

- The error is cleared if the operation mode is changed to other than reference position return before the axis performs reference position return.
- The error is cleared when the tool return is completed.
- The error is cleared if reset 1 is input or the emergency stop button is pushed.

M01	R-pnt ret invld at abs pos alm	0024

Details

[R-pnt ret invld at abs pos alm]

A zero point return signal was input during an absolute position detection alarm.

[Aux ax R ret invld at abs alm]

Reference position return was executed during an absolute position alarm.

Remedy

[R-pnt ret invld at abs pos alm]

- Reset the absolute position detection alarm, and then perform zero point return.

[Aux ax R ret invld at abs alm]

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M01	R-pnt ret invld at zero pt ini	0025

Details

[R-pnt ret invld at zero pt ini]

A zero point return signal was input during zero point initialization of the absolute position detection system.

[Aux ax R ret invld at ini]

Reference position return was executed during absolute position initial setting.

Remedy

[R-pnt ret invld at zero pt ini]

- Complete zero point initialization, and then perform zero point return.

[Aux ax R ret invld at ini]

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M01	High-accuracy skip disabled	0028

Details

The drive unit's hardware or software does not conform to the high-accuracy skip.

Remedy

- The software or hardware does not conform to the function. Contact service center.

M01	Hi-ac skip coord retrieval err	0029

Details

Failed to retrieve the skip coordinate value from the drive unit.

Remedy

- Check the wiring.
- Check the parameters.

M01	Now skip on	0030

Details

The "skip input" signal remains enabled when the operation has shifted from skip retract to measurement.

Remedy

- Increase the skip retract amount.

M01	No skip	0031

Details

Even though the 1st skip was to the correct position, the 2nd skip could not be found.

Remedy

- Check whether the measurement target has moved.

M01	Rtn dir err in manual measure	0033

Details

Return direction in manual measurement is the opposite of the parameter setting.

Remedy

- Check the setting of the parameter "#2169 Rtn dir err in manual measure" (Return direction in manual measurement).
- Move the axis manually in the direction to a safe position, then reset.

M01	No specifications	0036

Details

The specification is not supported.

Remedy

- Check the specifications.

M01	Chopping axis R-pnt incomplete	0050

Details

Chopping mode has been entered while the chopping axis has not completed reference position return.
All axes interlock has been applied.

Remedy

- Reset the NC or disable the "chopping" signal, and then carry out the reference position return.

M01	Synchronous error excessive	0051

Details

The synchronization error of the master and slave axes exceeded the allowable value under synchronous control.
A deviation exceeding the synchronization error limit value was found with the synchronization deviation detection.

Remedy

- Select the correction mode and move one of the axes in the direction in which the errors are reduced.
- Check the parameter "#2024 synerr".
- Increase the allowable value or reset it to "0" (check disabled).
- When using simple C-axis synchronous control, set the contents of the R2589 register to "0".

M01	No spindle select signal	0053

Details

Synchronous tapping command was issued when the spindle select signals (SWS) for all spindles were OFF in the multiple-spindle control II.

Remedy

- Turn ON the spindle select signal (SWS) responding to the tapping spindle before performing the synchronous tapping command.

M01	No spindle serial connection	0054

Details

Synchronous tapping command was issued when the spindle that the spindle select signal (SWS) was ON was not serially connected in the multiple-spindle control II.

Remedy

- Make sure the spindle select signal (SWS) for the responding spindle is ON.
- Consider the machine construction when issuing the command.

M01	Spindle fwd/rvs run para err	0055

Details

Asynchronous tapping command was issued when M code of the spindle forward/reverse run command, set by the parameter "#3028 sprcmm", was one of the followings in the multiple-spindle control II.

- M0, M1, M2, M30, M98, M99, or M198
- M code No. that commands to enable/disable the "macro interrupt" signal

Remedy

- Correct the parameter "#3028 sprcmm" (Tap cycle spindle forward run/reverse run M command) setting.

M01	Tap pitch/thread number error	0056

Details

The command for the pitch or the number of threads is not correct in the synchronous tapping command of the multiple-spindle control II.
The pitch is too small for the spindle rotation speed.
Thread number is too large for the spindle rotation speed.

Remedy

- Correct the pitch, number of threads or rotation speed of the tapping spindle.

M01	Wait for tap retract	0057

Details

The axis travel command is interlocked as the tap retract is being enabled.

Remedy

- If tapping is necessary, perform tapping retract in advance. However, tapping retract is not allowed during cycle start operation. Carry out tapping retract after resetting.
- If tapping is not necessary, cancel the tap retract enabled condition by the tap retract enabled cancel signal.

M01	Sync tap slave spindle error	0059

Details

The spindle synchronized with the slave axis (spindle set in parameter "#2713 stap_slave_sp_no") is not connected.

Remedy

- Set parameter "#2713 stap_slave_sp_no" to the spindle number of one of the connected spindles.
- Program the commands in accordance with the machine configuration.

M01	Handle ratio too large	0060

Details

The handle ratio is too large for the handle feed clamp speed.
(The handle feed clamp speed changes according to the rapid traverse rate, external feedrate, maximum speed outside the soft limit range and etc. (or external deceleration speed when external deceleration is valid))

Too large handle ratio means the ratio with which the machine is unable to move the distance of [iunit * handle ratio] within 0.1ms at the clamp speed.

Example)

When iunit=B (0.001mm) and clamp speed cs (mm/min)

The operation error (M01 0060) will occur if

ratio M>cs/60(s)*0.1(ms)/0.001(mm)=cs*10/6.

Remedy

- Change the settings of the handle feed clamp speed or the handle ratio.

M01	R-pos offset value illegal	0065

Details

At the start of reference position initial setting, the parameter "#2034 rfpofs" (Distance-coded reference position detection offset) is not set to "0".

Remedy

- Set the parameter "#2034 rfpofs" to "0", then turn the power ON again to perform the reference position initial setting.

M01	R-pos scan distance exceeded	0066

Details

Reference position could not be established within the maximum scan distance.

Remedy

- Check the scale to see if it has dirt or damage.
- Check if the servo drive unit supports this function.

M01	Illegal op in wk instl err cmp	0070

Details

One of the following operations was attempted during workpiece installation error compensation.

- Manual interruption
- Automatic operation handle interruption
- MDI interruption
- PLC interruption

Remedy

- Return the operation mode to the original mode to remove the cause. During workpiece installation error compensation, manual interruption, automatic operation handle interruption, MDI interruption, PLC interruption, etc. is not allowed.

M01	Illegal op in dia/rad select	0095

Details

An axis used in the following functions is also used as an axis to perform diameter/radius designation selection.

- Chopping
- Synchronization control

Or diameter/radius designation selection (G10.9) was issued to the axis in the above mode.

Remedy

- Check the program.
- Do not carry out the following functions using an axis to perform diameter/radius designation selection.
  - Chopping
  - Synchronization control

M01	No operation mode	0101

Details

[No operation mode]

No operation mode

[Aux ax no operation mode]

The operation mode is not designated, or the operation mode was changed during axis movement.

Remedy

[No operation mode]

- Check for any broken wires in the input mode signal wire.
- Check for any failure of the MODE SELECT switch.
- Correct the sequence program.

[Aux ax no operation mode]

- Correctly designate the operation mode.

M01	Cutting override zero	0102

Details

The "cutting feed override" switch on the machine operation panel is set to"0".
The override was set to "0" during a single block stop.

Remedy

- Set the "cutting feed override" switch to a value other than "0" to clear the error.
- If the "cutting feed override" switch has been set to a value other than "0", check for any short circuit in the signal line.
- Correct the sequence program.

M01	External feed rate zero	0103

Details

[External feed rate zero]

The MANUAL FEEDRATE switch on the machine operation panel is set to zero when the machine is in the jog mode or automatic dry run mode.

The "Manual feedrate B speed" signal is set to zero during the jog mode when Manual feedrate B is valid.

The "Each axis manual feedrate B speed" is set to zero during the jog mode when Each axis manual feedrate B is valid.

[Aux ax feedrate 0]

The feedrate set in the operation parameter is zero.

Or the override is enabled but is set to zero.

Remedy

[External feed rate zero]

- Set the MANUAL FEEDRATE switch to a value other than "0" to release the error.
- If the MANUAL FEEDRATE switch has been set to a value other than "0" check for any short circuit in the signal line.
- Correct the sequence program.

[Aux ax feedrate 0]

- Set a value other than zero in the feedrate setting or override value.

M01	F 1-digit feed rate zero	0104

Details

The F1-digit feedrate has been set to "0" when the F1-digit feed command was executed.

Remedy

- Set the F1-digit feedrate (the parameter "#1185 spd_F1" (F1 digit feedrate F1) to "#1189 spd_F5" (F1 digit feedrate F5)).

M01	Spindle stop	0105

Details

The spindle stopped during the synchronous feed/thread cutting command.

Remedy

- Rotate the spindle.
- If the workpiece is not being cut, start dry run.
- Check for any broken wire in the spindle encoder cable.
- Check the connections for the spindle encoder connectors.
- Check the spindle encoder pulse.
- Correct the program. (commands and addresses)

M01	Handle feed ax No. illegal	0106

Details

The axis, designated at handle feed, is out of specifications.
No axis has been selected for handle feed.
Multiple axes in a part system are allocated to a handle.

Remedy

- Check for any broken wires in the handle feed axis selection signal line.
- Correct the sequence program.
- Check the number of axes in the specifications.
- Check the axis allocation to the handle.

M01	Spindle rotation speed over	0107

Details

Spindle rotation speed exceeded the axis clamp speed during the thread cutting command.

Remedy

- Lower the commanded rotation speed.

M01	Fixed pnt mode feed ax illegal	0108

Details

The axis, designated in the manual arbitrary feed, is out of specifications.
The feedrate in manual arbitrary feed mode is illegal.

Remedy

- Check for any broken wires in the axis selection signal line or the feedrate line for the manual arbitrary feed mode.
- Check the specifications for the manual arbitrary feed mode.

M01	Block start interlock	0109

Details

An interlock signal has been input to lock the block start.

Remedy

- Correct the sequence program.

M01	Cutting block start interlock	0110

Details

An interlock signal has been input to lock the cutting block start.

Remedy

- Correct the sequence program.

M01	Restart switch ON	0111

Details

Restart switch has been turned ON and manual mode has been selected before the restart search is completed.

Remedy

- Search the block to restart.
- Turn the restart switch OFF.

M01	Program check mode	0112

Details

The cycle start button was pressed during program check or in program check mode.

Remedy

- Press the reset button to cancel the program check mode.

M01	Cycle st. in buffer correct	0113

Details

The cycle start button was pressed during buffer correction.

Remedy

- Press the cycle start button after the buffer correction is completed.

M01	In reset process	0115

Details

The cycle start button was pressed during resetting or tape rewinding.

Remedy

- When rewinding the tape, wait for the winding to end, or press the reset button to stop the winding, and then press the cycle start button.
- During resetting, wait for the resetting to end, and then press the automatic start button.

M01	Playback not possible	0117

Details

The playback switch was turned ON during editing.

Remedy

- Cancel the editing by pressing the input or previous screen key before turning ON the playback switch.

M01	Turn stop in normal line cntrl	0118

Details

The turning angle at the block joint exceeded the limit during normal line control.
In normal line control type I:
The parameter "#1523 C_feed" (Normal line control axis turning speed) has not been set.
In normal line control type II:
When turning in the inside of the arc, the set value for the parameter "#8041 C-rot. R" is larger than the arc radius.

Remedy

- Correct the program.
- Correct the parameter "#1523 C_feed" (Normal line control axis turning speed) setting.
- Correct the parameter "#8041 C-rot. R" setting.

M01	Reverse run impossible	0119

Details

Any of the following conditions are occurring.
a) There is no block to run backward
b) Eight blocks without a travel command continued

Remedy

- Execute forward run to clear the alarm.
- Reset to clear the alarm.

M01	In synchronous correction mode	0120

Details

The synchronous correction mode switch was pressed in non-handle mode.

Remedy

- Select the handle or manual arbitrary feed mode.
- Turn OFF the correction mode switch.

M01	No synchronous control option	0121

Details

The synchronous control operation method was set (with R2589) while no synchronous control option was provided.

Remedy

- Set "0" for "synchronous control operation method".

M01	Computer link B not possible	0123

Details

Cycle start was attempted before resetting was completed.
Computer link B operation was attempted at the 2nd or further part system in a multi-part system.

Remedy

- Perform the cycle start after resetting has been completed.
- Set "#8109 HOST LINK" to "0" and then set to "1" before performing the cycle start.
- Computer link B operation cannot be performed at the 2nd or further part system in a multi-part system.

M01	X/Z axes simultaneous prohibit	0124

Details

The basic axis corresponding to the inclined axis was started simultaneously in the manual mode while the inclined axis control was valid.

Remedy

- Turn the inclined axis and basic axis start OFF for both axes. (This is also applied for manual/automatic simultaneous start.)
- Disable the basic axis compensation, or command it to axes one by one.

M01	Rapid override zero	0125

Details

- The RAPID TRAVERSE OVERRIDE switch on the machine operation panel is set to "0".
- Override was set to "0" during single block stop.

Remedy

- Set the RAPID TRAVERSE OVERRIDE switch to a value other than "0" to clear the error.
- If the RAPID TRAVERSE OVERRIDE switch has been set to a value other than "0", check for any short circuit in the signal line.
- Correct the sequence program.

M01	Program restart machine lock	0126

Details

Machine lock was applied on the return axis being manually returned to the restart position.

Remedy

- Cancel the machine lock and resume the operation.

M01	Rot axis parameter error	0127

Details

- Orthogonal coordinate axis name does not exist.
- Rotary axis name does not exist.
- A duplicate name is used for the designated orthogonal coordinate axis.
- The number of axes that were selected to change tool length compensation along the tool axis amount exceeds the maximum number of axes.
- The designated orthogonal coordinate axis name is the same as the rotary axis name.
- There is an invalid rotary axis setting in the rotary axis configuration parameters.
- A function that cannot be commanded when the spindle end offset is set was commanded.

Remedy

- Correct the rotary axis configuration parameters.

M01	Restart pos return incomplete	0128

Details

Cycle start was performed with an axis whose return to the restart position was not complete.

Remedy

- Perform restart position return manually.
- Validate the parameter "#1302 AutoRP" (Automatic return by program restart), then execute cycle start.

M01	PLC interruption impossible	0129

Details

After the cycle start, the "PLC interrupt" signal was turned ON during buffer correction, program restart, arbitrary reverse run, tool retract and return, high-speed high-accuracy control II, NURBS interpolation or single block stop at the middle point of G28/G29/G30.

Remedy

- By turning OFF the "PLC interrupt" signal, or by resetting the NC the error can be cancelled.

M01	Restart posn return disabled	0130

Details

Restart position return was attempted in a mode where the return is disabled.

Remedy

- Correct the program restart position.

M01	Zero point return interruption	0131

Details

Compound type fixed cycle program was interrupted with manual zero point return, and cycle start was carried out without carrying out reset.

Remedy

- Cancel the program execution by reset.

M01	Excessive no. of reverse block	0133

Details

During the reverse run in arbitrary reverse run, any one of the part systems reverse-ran for 20 blocks.

Remedy

- This error is cancelled by forward run.

M01	Illegal mode in prg check mode	0134

Details

Any of the following modes has been turned ON during the manual arbitrary reverse run mode.

- MDI interruption mode
- Manual/automatic simultaneous valid (MAE1 to 8)
- Arbitrary feed mode (PTP)
- Arbitrary reverse control mode (RVMD)
- Manual speed command enabled
- High-speed simple program check mode (SMLK)

Remedy

- The error can be cancelled by turning OFF the error-causing mode.

M01	Too many active axes per sys	0135

Details

High-accuracy control has been executed in a part system which has 9 or more enabled control axes, except for a slave or synchronized axis.

Remedy

- Reduce the number of enabled control axes (except for a slave axis and synchronized axis) of the part system to 8 or less through synchronous control, control axis synchronization between part systems, or mixed control before executing high-accuracy control.

M01	Pre-intrpl variable accel err	0136

Details

- The parameter "#12060 VblAccPreInt" (Variable-acceleration pre-interpolation acceleration/deceleration ON) is set to "1" and the parameter "#8090 SSS ON" is set to "0".

Remedy

- To enable the variable acceleration pre-interpolation acceleration/deceleration, set the parameter "#8090 SSS ON" to "1".
  To disable the variable acceleration pre-interpolation acceleration/deceleration, set the parameter "#12060 VblAccPreInt" to "0".

M01	Unable to start automatic mode	0137

Details

- Although start of automatic operation is not allowed after a machine groupwise alarm stop ("#1472 mgralmrestart"=0), start of automatic operation has been attempted after the said alarm stop.

Remedy

- Remove the cause of the stop by alarm.
- Set the parameter "#1472 mgralmrestart" to "1".

M01	Tool data sorting in progress	0138

Details

Cycle start or graphic check has been attempted during sorting of tool management data.

Remedy

- Execute cycle start after the tool data sorting is completed.
- Execute graphic check after the tool data sorting is completed.

M01	Tolerance control invalid	0139

Details

The parameter "#12066 Tolerance ctrl ON" is set to "1", although "#8090 SSS ON" is "0".

Remedy

- To enable the tolerance control, set "#8090 SSS ON" to "1".
  To disable the tolerance control, set "#12066 Tolerance ctrl ON" to "0".

M01	Start err in G68/G68.1 manu fd	0140

Details

Axis activation has been attempted with any of the following operations while manual feed is enabled in 3-dimensional coordinate conversion or Coordinate rotation by program.

- Manual tool length measurement
- Workpiece position measurement
- Tool retract and return

Remedy

- Turn OFF the "3-dimensional coordinate conversion or Coordinate rotation by program: Coordinate switch for manual feed" signal.

M01	Mult ax in G68/G68.1 manual fd	0141

Details

Two or more of the three base axes have been activated simultaneously while manual feed is enabled in 3-dimensional coordinate conversion or Coordinate rotation by program.

Remedy

- Disable the manual feed in 3-dimensional coordinate conversion or Coordinate rotation by program, or activate one axis at a time.

M01	Ill manualmode select in 3Dcnv	0145

Details

The following unselectable manual mode was selected during 3-dimensional coordinate conversion.

- manual reference position return

Remedy

- Cancel the 3-dimensional coordinate conversion modal.

M01	Start err in PRM rot manu feed	0146

Details

Axis start has been attempted with any of the following operations while the coordinate rotation by parameter for manual feed is enabled.

- Tool retract and return
- Manual tool length measurement
- Manual skip

Remedy

- Turn OFF the "Coordinate rotation by parameter: Coordinate switch for manual feed" signal.

M01	Multiaxes in RRM rot manu feed	0147

Details

Two or more of the three basic axes were started at a time for the coordinate rotation by parameter for manual feed.

Remedy

- Disable the coordinate rotation by parameter for manual feed or start one axis at a time.

M01	Unable to start automatic mode	0149

Details

The searched program has been altered due to the change made to the setting of the location of the program.

An attempt is being made to run a program that is not the one operation search was carried out for.

Remedy

- Perform operation search for the program.

M01	Chopping override zero	0150

Details

The override became "0" in the chopping operation.

Remedy

- Check the chopping override (R2503).
- Check the rapid traverse override (R2502).

M01	Command axis chopping axis	0151

Details

A chopping axis movement command was issued from the program during the chopping mode. (This alarm will not occur for the command with the movement amount "0".)
(All axes interlock state will be applied.)

Remedy

- Press the reset button or turn OFF the "chopping" signal. When the "chopping" signal is turned OFF, the axis returns to the reference position and performs the movement command in the program.

M01	Bottom dead center pos. zero	0153

Details

The bottom dead center position is set to the same position as the upper dead center position.

Remedy

- Correct the bottom dead center position.

M01	Chopping disable for handle ax	0154

Details

Chopping has been attempted while the chopping axis is selected as the handle axis.

Remedy

- Select an axis other than the chopping axis as the handle axis, or start chopping after changing the mode to the other mode.

M01	Dir cmnd mode invalid	0157

Details

- The drive unit's software or hardware does not conform to the direct command mode.
- Inclined axis control is active.
- Control axis synchronization between part systems is active.
- Control axis superimposition was activated during direct command mode.

Remedy

- The software or hardware does not conform to the function. Contact service center.
- Turn the inclined axis control valid signal OFF.
- Turn the synchronous control request signal OFF.
- Turn the superimposition control request signal OFF.

M01	Dir cmnd mode restart invalid	0158

Details

- Cycle start was carried out without reset after the retract in direct command mode.

Remedy

- Finish the machining for now by resetting the NC.

M01	OMR-CC invalid	0159

Details

When "#2678 OMRCC_valid" is set to "1", "#2139 omrff_off" is set to "1" or "#2313 SV113 SSF8/bit0" is set to "0".

Remedy

- To enable OMR-CC, set "#2139 omrff_off" to "0", and "#2313 SV113 SSF8/bit0" to "1".
- To disable OMR-CC, set "#2678 OMRCC_valid" to "0".

M01	No speed set out of soft limit	0160

Details

[No speed set out of soft limit]

For the axis for which the maximum speed in the outside of the soft limit range has not been specified, an attempt was made to return the axis from the outside of the soft limit range.

[Aux ax sta No. illegal]

The designated station number is exceeding the number of indexed divisions.

Remedy

[No speed set out of soft limit]

- Set the maximum speed for the outside of the soft limit range.
  (Parameter "#2021 out_f")
- Correct the soft limit range.
  (Parameters "#2013 OT-", "#2014 OT+", "#8204 OT-CHECK-N" and "#8205 OT-CHECK-P")

[Aux ax sta No. illegal]

- Correctly designate the station No.

M01	Aux ax R-pnt ret incomplete	0161

Details

Automatic/manual operation was started before reference position return was executed with the incremental system.

Remedy

- Execute the reference position return.

M01	Aux abs position initializing	0162

Details

The start signal was input while initializing the absolute position reference point.

Remedy

- Complete the absolute position reference point initialization.

M01	Aux ax abs position error	0163

Details

The start signal was input during an absolute position alarm.

Remedy

- Initialize the absolute position reference point and then fix the absolute position coordinates.

M01	Aux ax arbitrary positioning	0164

Details

The manual operation mode was started during the random positioning mode.

Remedy

- Turn the random positioning mode OFF before switching to the manual operation mode.

M01	Aux uneven index sta No. ilgl	0165

Details

The commanded station No. was higher than 20 or the number of indexing stations during uneven indexing.
The station No.0 was specified during uneven indexing.

Remedy

- Check the commanded station No. and the parameter "#12801 aux_station" setting.

M01	Aux axis changeover error	0166

Details

One of the following attempts was made on an axis that is switchable between NC axis and auxiliary axis.

- A command intended for an NC axis in manual mode was issued to an auxiliary axis.
- NC axis control select signal was turned OFF while the NC axis was in motion.
- NC axis control select signal was turned ON while the auxiliary axis was in motion.

Remedy

- If you wish to issue a command intended for an NC axis in manual mode, turn ON the NC axis control select signal so as to set the axis as an NC axis.
- Do not change NC axis control select signal while the axis is in motion.

M01	Aux ax torque limit value zero	0167

Details

Torque limit value of the parameter group to use is "0".

Remedy

- Check the torque limit value of the parameter group to use (#12814 aux_TL1, #12824 aux_TL2, #12834 aux_TL3, or #12844 aux_TL4).

M01	Ill. op during T tip control	0170

Details

Illegal operation was attempted during tool tip center control.

Remedy

- Change the operation mode to the previous one and restart.

M01	Illegal operation in TCPC	0172

Details

An illegal operation was performed during tool cutting point control mode.

Remedy

- Revert the system to the state prior to the operation and restart the CNC.

M01	Illegal op in spline interpol2	0180

Details

Any of the following operations was performed during spline interpolation 2.

- Change to manual mode
- Change to MDI mode
- PLC interruption

Remedy

- Return to the original operation mode, and then remove the error cause.

M01	Ill. op in 2D barcode engrave	0181

Details

An illegal operation was attempted during the two-dimensional barcode engraving cycle.

Remedy

- Switch the operation back to the previous mode, and remove the error cause.

M01	Illegal op in text engraving	0182

Details

An illegal operation was attempted during Text engraving cycle.

Remedy

- Change the operation mode back to the previous one, and remove the cause of the error.

M01	Illegal OP in tilted face cut	0185

Details

Any of the following illegal operations was attempted during inclined surface machining mode.

- Manual interrupt
- Handle interrupt in automatic operation
- MDI interrupt
- PLC interrupt
- Arbitrary reverse run

Remedy

Switch the operation mode back to the previous to remove the cause of this failure.

- During inclined surface machining mode, it's impossible to perform manual interrupt, handle interrupt in automatic operation, MDI interrupt, PLC interrupt, etc.

M01	Invalid axis configuration	0186

Details

The operation you input cannot be executed under the axis configuration of the part system.

- 3D manual feed was attempted.
- The rotation center error compensation has been turned ON.
- The rotary axis is set to the spindle mode of the spindle position control.
- An unavailable function command was given when "Select specifications of rotation direction parameter" is enabled (#1450/bit3 = 1), and the left-hand screw direction is selected by the rotation direction parameter (#7923, #7933, #7943, #7953 = 1).

Remedy

- Check the rotary axis configuration parameters.
- Correct the axis configuration of the part system.
- Reset the NC or remove the error cause to cancel the alarm.

M01	Rotary axis para unswitchable	0187

Details

The rotary axis configuration parameter switch request signal has turned ON for a part system where any of the following modes is active.

- Tool center point control mode
- Tool cutting point control mode
- Inclined surface machining mode
- Workpiece installation error compensation mode
- Tool length compensation along the tool axis mode
- Simple inclined surface machining mode
- 3-dimensional tool radius compensation mode
- 3-dimensional manual feed
- Tool handle feed & interruption
- R-Navi mode

Remedy

- Cancel the alarm by CNC reset or by turning OFF the rotary axis configuration parameter switch request signal.

M01	Illegal op in MCR coord TF	0188

Details

An illegal operation was performed during motion control release (coordinate transformation).

Remedy

- Roll back the operation that caused the error and restart the NC.

M01	MCR calculation error	0189

Details

Illegal coordinate transformation during motion control release (coordinate transformation).

Remedy

- Report this error to the machine tool builder.

M01	Interference check disabled	0200

Details

3D machine interference check is disabled. This alarm is output to NC alarm 5.

Remedy

- Enable all the interference check settings.
- If there is any axis which has not completed zero point return, establish the zero point first.

M01	Machine interference 1	0201

Details

It was judged that an interference occurred in the No.1 step interference check and caused a deceleration stop. When machine interference is detected, the interfered part is highlighted (yellow/red) and the part's name is displayed on the 3D monitor's model display.

Remedy

- Move the axis in a direction which does not cause interference.
- Press RESET to cancel the alarm.
- (In manual operation)
  You can move the axis in the same travel direction as before the interference.
  But the axis movement is done using the 2nd step interference check distance.

M01	Machine interference 2	0202

Details

It was judged that an interference occurred in the No.2 step interference check and caused a deceleration stop. When machine interference is detected, the interfered part is highlighted (red) and the part's name is displayed on the 3D monitor's model display.

Remedy

- Move the axis in a direction which doesn't cause interference.
- Press RESET to cancel the alarm.

M01	Tool interfere. check disabled	0205

Details

Tool interference check has been disabled.

This alarm is output to the NC alarm 5.

Remedy

- Turn ON the [Tool check] menu of 3D monitor.
- Set the tool data.
- If you prevent an alarm from being output while the tool interference check is disabled, set "0" in "#11100 3D_MChk_ToolAlm".

M01	Too many simul. control axes	0211

Details

The given command has caused any axis other than those commanded to move. So the total number of axes to move has exceeded the maximum number of simultaneous contour control axes.

Remedy

- Check the maximum number of simultaneous contour control axes of your NC.
- Check the machining program to make sure the total number of axes to move will not exceed the maximum number of simultaneous contour control axes.

M01	Changing prg format disabled	0215

Details

The PFCHR signal has been switched (from ON to OFF or OFF to ON) during automatic operation.

Remedy

- Use G188 to change the program format during automatic operation.

* The warning display is cancelled by changing PFCHR back to the previous state.

M01	Rapid block overlap inv	0216

Details

The following parameters are enabled while there is no option for Rapid traverse block overlap.

- "#1442 G0ol" (Enable G00 rapid traverse block overlap)
- "#1443 G28ol" (Enable G28 rapid traverse block overlap)
- "#12056 I_G0ol" (Enable G00 initial rapid traverse block overlap)

Remedy

- Check the specification.
- Disable the parameter.

M01	OP invld during MTB macro prep	0217

Details

NC performed automatic operation while preparing MTB macro.

Remedy

- Execute the program again after reset.

M01	Too many axes in 3D interf chk	0220

Details

You have set 13 or more axes as the objectives of 3D machine interference check.

Remedy

- Correct the setting of "#2673 3D_MChk_Ax".

M01	Invalid cmd in 3D interf check	0221

Details

The commanded function is invalid during 3D machine interference check.

Remedy

- Disable interference check before the command.

M01	Multi ax for 3D manual feed	0230

Details

More than one axis was designated in manual mode while the 3-dimensional manual feed was valid.

Remedy

- Command the manual feed to each axis one by one.

M01	3D manual feed coord sys err	0231

Details

- More than one of the three bits for selecting hypothetical coordinate system was turned ON.
- Virtual coordinate system was selected while the 3-dimensional manual feed was invalidated by the parameter setting.
- The manual tool length measurement or workpiece position measurement is being attempted.

Remedy

- Check the sequence program.
- Enable 3-dimensional tool radius compensation (set the parameter "#7912 NO_MANUAL" to "0").
- Finish the manual tool length measurement or workpiece position measurement.

M01	Illegal op in 3D tool R comp	0232

Details

An illegal operation (such as manual interrupt) was attempted during 3-dimensional tool radius compensation (tool vertical direction compensation).

Remedy

- Operations such as manual interrupt are disabled while 3-dimensional tool radius compensation (tool vertical direction compensation) is being performed.

M01	Coord select w/o 3D man. Feed	0233

Details

Hypothetical coordinate system has been selected although 3D manual feed is not included in the specifications.

Remedy

Turn OFF all the 3D manual feed coordinate system selection signals (including MJCT).

M01	No spec: Rot center error comp	0240

Details

No option of rotation center error compensation is found.

Remedy

- Check the specifications.

M01	Rot center err comp incorrect	0241

Details

- The calculated compensation amount was excessive, thus the amount was clamped to +/- 1mm.
- Compensation for angle deviation was executed although the preset tool length is not in the height-axis direction.

Remedy

- Check the values of the position deviation and the angle deviation.
- Check the tool length value.

M01	Unable to start measuring err	0245

Details

You executed cycle start while measurement of rotation center error or workpiece installation error is invalid.

Remedy

Execute cycle start for one part system that is ready for measurement.
Activate memory mode before executing cycle start.

M01	Invalid op mode in err measure	0246

Details

You switched the operation to any mode other than memory during measurement of rotation center error or workpiece installation error.

Remedy

Switch the operation to memory mode.

Manual interruption, automatic operation handle interruption, MDI interruption, etc. are invalid during error measurement.

Execute cycle start after completion or cancel of the measurement.

M01	Machining surface operation disabled	0250

Details

Machining surface operation (selection, indexing or cancel) was attempted while the operation is disabled.

Remedy

- Cancel the other modes so that the inclined surface machining command (G68.2), tool axis direction control (G53.1) and the inclined surface machining cancel command (G69) can be issued.
- Wait until the axes stop completely (until the smoothing for all axes reaches zero).
- Perform operation search for machining programs.

M01	Axs travel n/a in manual index	0251

Details

Moving a rotary axis was attempted during manual machining surface indexing, when a manual operation mode other than handle mode was selected.

Remedy

- Change the operation mode to a handle mode before carrying out the manual surface indexing.

M01	Tool length compensation amt 0	0252

Details

The tool length compensation amount for performing the R-Navi indexing type 2 is 0.

Remedy

- Set the tool length compensation amount for performing the indexing type 2 to a value other than 0.

M01	Feat coord ill w/ multi-handle	0253

Details

Manual feed feature coordinate system was selected while two or more handles were ON.

Remedy

- Manual feed on a feature coordinate system is disabled while two or more handles are enabled (*).
- Press [Manual coord] and select the machine coordinate system.
- Reduce the number of enabled handles (*) to one.
- (*) An enabled handle means the handle for which "Nth handle valid" signal (HSnS) is ON.

M01	No spec: Spatial error comp	0260

Details

- The spatial error compensation option is not available.
- Spatial error compensation cannot be executed while "Select specifications of rotation direction parameter" is enabled (#1450/bit3 = 1) and "Rotation direction" is set to "left-hand screw direction" (#7923, #7933, #7943 or #7953 = 1).
- Spatial error compensation cannot be executed while "#11484 Spec_valid/bit1" (Enable Spatial error compensation) is "0". (M80VW/M80V TypeA)

Remedy

- Check the specifications.

M01	Spatial error comp excessive	0261

Details

The compensation amount is outside the setting range.
Due to the calculation of the excessive amount, it is clamped by +/-1mm.

Remedy

Set the compensation amount within the setting range.

M01	Auto backlash adjust illegal	0270

Details

- A measurement condition adjustment or backlash adjustment was attempted to an axis with which automatic backlash adjustment is impossible.
- A measurement condition adjustment or backlash adjustment was attempted even though all the axes had not reached the 1st reference position.
- The operation mode is other than memory mode.
- The slave axis of synchronous control is selected as adjustment axis.
- An attempt has been made to start an adjustment by cycle start.

Remedy

- Check the adjustment axis.
- Start the adjustment after all the axes return to the 1st reference position.
- Check the operation mode.
- Select the master axis of synchronous control as adjustment axis when adjusting the slave axis.
- Start the adjustment by automatic backlash adjustment start signal.

M01	Operating auto backlash adjust	0271

Details

An illegal operation was attempted during measurement condition adjustment or backlash adjustment.

Remedy

- Continue the operation after canceling the measurement condition adjustment and backlash adjustment.

M01	APLC password mismatch	0280

Details

The APLC authentication password is inconsistent.

Remedy

- Contact the machine tool builder.

M01	High-cycle sampling disabled	0290

Details

- The drive unit's hardware or software does not conform to the high-cycle sampling mode.
- High-cycle sampling data are not used even when high-cycle sampling has been set.
- High-cycle sampling was attempted while the high-cycle sampling axis was in operation.
- High-cycle sampling was attempted during speed monitor mode.
- High-cycle sampling was attempted during executing any of the following operations: Dog type zero point return, absolute position setting, synchronous tapping, spindle synchronization, hobbing and tool spindle synchronization IC.
- High-cycle sampling was attempted while multi-step speed monitor is active.
- High-cycle sampling was attempted while constant speed monitor is active.
- High-cycle sampling was attempted while real-time data sampling is active.

Remedy

- The software or hardware does not conform to the function. Contact service center.
- Set data for high-cycle sampling.
- Execute high-cycle sampling after stopping the high-cycle sampling axis.
- Execute high-cycle sampling after canceling the speed monitor mode.
- Execute high-cycle sampling after stopping the function currently executed.
- Execute high-cycle sampling after disabling the multi-step speed monitor.
- High-cycle sampling cannot be executed while constant speed monitor is active.
- Execute high-cycle sampling after stopping the real-time data sampling.

M01	N/A during high-cycle sampling	0291

Details

- An attempt to activate "Speed monitor mode" was made during the high-cycle sampling mode.
- An attempt to change the gear signal was made during the high-cycle sampling mode.
- An attempt to execute spindle orientation was made during the high-cycle sampling mode.
- Spindle detach was attempted during the high-cycle sampling mode.
- Any of the following operations was attempted during the high-cycle sampling mode: Dog-type zero point return, absolute position setting, spindle/C axis changeover, synchronous tapping, spindle synchronization, hobbing, or tool spindle synchronization IC.

Remedy

- Change the speed monitor mode signal back, finish high-cycle sampling, and then select the speed monitor mode.
- Change the gear signal back, finish high-cycle sampling, and then change the gear.
- Change the spindle orientation signal back, finish high-cycle sampling, and then execute orientation.
- Change the spindle detach signal back, finish high-cycle sampling, and then carry out spindle detachment.
- Execute the operation after terminating high-cycle sampling.

M01	Invalid cmd during param write	0292	(Axis name)

Details

A function which is unavailable during parameter write was commanded.

- Spindle/C axis changeover
- Speed observation mode signal ON
- High-speed synchronous tapping
- PLC axis indexing

Remedy

- Cancel the write of the parameter.
- Command the following functions after the parameter write is completed.
  - Spindle/C axis changeover
  - Speed observation mode signal ON
  - High-speed synchronous tapping
  - PLC axis indexing

M01	SP orient. Z detection error	0301

Details

Z phase detection has failed to be completed during execution of proximity-switch orientation.

Remedy

- Check for a failure of the proximity switch.

M01	Thread cutting axis in motion	0310

Details

- While the lead axis was moving, the thread groove position command (G123) from which the address I was omitted has been issued.
- While the spindle was rotating, the thread groove position command (G123) from which the address C was omitted has been issued.

Remedy

- Issue the command G123 from which the address I or the address C is omitted while the lead axis or the spindle is stopped.

M01	Mnl Arb Rvs Run NumPartSystems	0333

Details

There is no part system for which the manual arbitrary reverse run part system selection (YC84) signal is ON.

Remedy

- Ensure that the manual arbitrary reverse run part system selection (YC84) signal is ON for at least one part system.

M01	Cmd invalid manual arb rev run	0334

Details

When "#1753 cfgPR03/bit1" (Manual arbitrary reverse run: operation per part system) is set to "1", the following command was issued between the part system that selected Manual arbitrary reverse run and the part system that did not select Manual arbitrary reverse run.

- Mixed control
- Arbitrary axis exchange control
- Control axis synchronization between part systems
- Control axis superimposition
- Two-part system simultaneous thread cutting

Remedy

- Turn OFF "Manual arbitrary reverse run mode (Y73C)" signal, then perform forward run to the start position for reverse run if necessary, then end "Manual arbitrary reverse run mode". The operation is restarted when "Manual arbitrary reverse run mode ON (X715)" signal is turned OFF.
- Reset the part system with the "Manual arbitrary reverse run" is enabled, and end the automatic operation.

M01	BiSS encoder comm error 1	0350

Details

Communication with BiSS encoder failed.

Remedy

- Check parameters "#11376 BiSS_enc_rate" through "#11380 BiSS_enc1_CRC_init".

M01	BiSS encoder comm error 2	0351

Details

Communication with BiSS encoder failed.

Remedy

- Check the cable connected with BiSS encoder.

M01	BiSS encoder comm error 3	0352

Details

Communication with BiSS encoder failed.

Remedy

- Check the cable connected with BiSS encoder.

M01	RTDS communication error	0360

Details

Communication error occurred between the control unit and the collector PC, and the sampling data cannot be sent.

Remedy

- Check the communication with the collector PC.

M01	Unable to start BrakeWear Diag	0370

Details

The conditions for starting brake wear diagnostics are not met when the brake wear diagnostics start signal (BWDSTm) is turned on.

The cause preventing brake wear diagnostics from starting is displayed.
(If there is more than one cause, the cause with the smallest number is displayed.)

0001: part systems in automatic operation

0002: not in in-position

0003: in servo OFF state

0004: in current limit

0005: synchronous control secondary axis

0006: in superimposition control

0007: in arbitrary axis exchange control

0008: in mixed control

0009: invalid brake wear diagnostics parameters

000A: reference position establishment incomplete

000B: meeting exclusion condition of brake wear diagnostics

000C: secondary axis brake wear diagnostics invalid

000D: secondary axis unable to start brake wear diagnostics

000E: secondary axis in single method

1001: unable to start due to Smart Safety observation

1002: PLC axis unable to start brake wear diagnostics

1003: in brake wear diagnostics alarm reset ON

1004: in coordinate rotation

1005: controlled as auxiliary axis

1007: inclined axis under inclined axis control

1008: multiple-axis synchronization control secondary axis

1009: secondary axis brake wear diagnostics invalid

100A: secondary axis unable to start brake wear diagnostics

100B: axis involved in control axis synchronization between part systems

Remedy

Check the conditions for starting brake wear diagnostics.

Brake wear diagnostics will be started after the cause of the alarm is resolved, and the brake wear diagnostics start signal is turned on again.

M01	BrakeWearDiagnostics (Warning)	0371

Details

The axis travel amount exceeded the allowable value during brake wear diagnostics.

The axis name of the axis where the error occurred is displayed.

Remedy

- Move the axis to a safe position with manual operation, turn off the power, and take corrective action for the brake. This warning is cleared upon successful completion of brake wear diagnostics, which is performed after the power is turned back on.
- This warning can be cleared by the brake wear diagnostics alarm reset signal. The brake wear diagnostics incomplete signal (BWDNFm) remains on.

M01	No spec: Brake wear diagnostics	0372

Details

Specifications do not include brake wear diagnostics.

Remedy

- Check the specifications.

M01	No brake wear diag response	0373

Details

There is no brake response when the drive unit internal control circuit method is selected.

Remedy

- Check the sequence program.
- Upgrade the software of the drive unit.
- The drive unit may have failed. Contact the service center.

M01	Stopped on NCAID diag-alarm	0380

Details

The CNC automatic operation is stopped due to an NCAID alarm.

Remedy

- Resolve the cause of NCAID alarm and then reset the CNC.
- Stop the spindle before cancelling this alarm.
  The spindle may start after the alarm is canceled.

M01	Illegal movement command during superimposition	1003

Details

- A machine command was issued to the superimposed axis.
- Reference position return was attempted on the superimposed axis.
- Skip command was issued to the reference or superimposed axis.
- Dog-type reference position return was attempted on the reference axis.

Remedy

- Correct the program.

M01	Superimposition command illegal	1004

Details

- Superimposition start command was issued to the axis which was in any of the following states.

  Synchronization control

  Milling interpolation

  Feed-forward thread cutting control

- Superimposition start command was issued to the axis which was under superimposition control.
- Superimposition start command was issued to an axis that belongs to the same part system as a reference or synchronized axis of control axis synchronization between part systems.

Remedy

- Correct the program.
- Turn OFF the feed-forward control request signal.

M01	G114.n command illegal	1005

Details

G114.n has been commanded during the execution of G114.n.
G51.2 has been commanded when G51.2 spindle-spindle polygon machining mode has been already entered at another part system.
Multiple spindle synchronization set command is illegal.

Remedy

- Command G113 to cancel the operation.
- Turn ON the "spindle synchronization cancel" signal (Y18B8: SPSYC) to cancel the operation.
- Command G50.2 to cancel the operation.
- Turn ON the "spindle-spindle polygon cancel" signal (YCD1) to cancel the operation.
- Correct the program.

M01	Spindle in-use by synchro tap	1007

Details

The spindle is being used in synchronized tapping.

Remedy

- Cancel the synchronized tapping.

M01	GB spindle synchro signal OFF	1014

Details

- A forward run, reverse run, orientation, synchronous tapping, spindle synchronization, tool spindle synchronization I, tool spindle synchronization II or C-axis servo ON command was issued to the reference spindle while the guide bushing spindle synchronization signal was OFF.
- The guide bushing spindle synchronization signal was turned OFF during a forward run, reverse run, orientation, synchronous tapping, spindle synchronization, tool spindle synchronization I, tool spindle synchronization II or C-axis servo ON command.
- Orientation was commanded during the "guide bushing spindle synchronization" signal ON with spindle zero point detection with proximity switch and turret indexing enabled.
- C axis servo ON was commanded during the "guide bushing spindle synchronization" signal ON with spindle C axis parameter change enabled.

Remedy

- Check the ladder program.
- Check the parameters.

M01	GB SP sync:Spindle type error	1015

Details

- An analog spindle is used for the master spindle or the guide bushing spindle.
- Spindle-mode servo is used for the master spindle or the guide bushing spindle.
- Turret gear change control valid spindle is used for the master spindle or the guide bushing spindle.

Remedy

- Check the parameters.
- Change the reference spindle or the guide bushing spindle to spindle drive unit.
- Change the reference spindle or the guide bushing spindle to turret gear change control invalid.

M01	GB SP sync:Phase mem sgnl ilgl	1021

Details

- The guide bushing spindle synchronization phase memory signal was turned ON while the master spindle or guide bushing spindle was rotating.
- The guide bushing spindle synchronization phase memory signal was turned ON while the guide bushing spindle synchronization signal was OFF.

Remedy

- Check the ladder program.

M01	GB SP sync:Phase set sgnl ilgl	1022

Details

- The guide bushing spindle synchronization phase alignment signal was turned ON while the master spindle or guide bushing spindle was stopped.

Remedy

- Check the ladder program.

M01	GB SP sync:Z phase not pass	1023

Details

- When the guide bushing spindle synchronization phase memory signal was ON, the master spindle or guide bushing spindle’s Z-phase was not passed.

Remedy

- Check the ladder program.

M01	Cmnd impsbl in spindle rtry ax	1024

Details

- A command to switch the spindle-mode rotary axis to the servo axis mode was given to the spindle for synchronous tapping or that for polygon machining.
- The spindle-mode rotary axis was assigned as synchronous tapping spindle or as polygon machining spindle while the rotary axis was in the servo axis mode.
- The spindle-mode rotary axis was assigned as reference spindle (master) or synchronized spindle (slave) of the spindle synchronization I/II or the tool spindle synchronization IA/IB/II.

Remedy

- Switch the motor control to the spindle-mode rotary axis mode.
- Reset the NC to cancel the error. Then correct the program.

M01	Other cmnd disabled in orient.	1025

Details

- Spindle superimposition control command has been given to an orientation mode spindle with the said spindle treated as either the basic or superimposed spindle.
- Orientation command has been given to the basic or superimposed spindle that is under spindle superimposition control.

Remedy

- Cancel the orientation mode.
- Use G113 or the "Spindle sync cancel" signal to cancel spindle superimposition.

M01	SP-C ax ctrl runs independntly	1026

Details

C axis mode command has been issued for polygon machining spindle.
C axis mode command has been issued for synchronized tapping spindle.
Polygon command has been issued for synchronized tapping spindle.
Spindle is being used as spindle/C axis.

Remedy

- Cancel the C axis command.
- Cancel the polygon machining command.
- Cancel the C axis with servo OFF.

M01	Thread recutting impossible	1027

Details

- The lead axis for thread recutting is not present in a thread recut enabled part system.
- A variable lead thread cut command has been given in the first thread cut block of thread recutting.

Remedy

- Thread recutting is not supported. Disable thread recutting or press RESET to cancel the alarm, and correct the program.

M01	Thread recutting data illegal	1028

Details

The lead axis and spindle for thread cutting are different from those predetermined.

Remedy

- Disable thread recutting or press RESET to cancel the alarm, and correct the settings of lead axis and spindle.

M01	Variable speed thread disabled	1029

Details

- "#8045 Varying spd thread" has been set to "1" to issue a variable speed thread cut command even though the function is not included in the specifications.
- A variable speed thread cut command has been given while the feed axis acceleration/deceleration type is soft acceleration/deceleration.
- A variable speed thread cut command has been given while the spindle encoder signal is not sent via serial input.
- A variable speed thread cut command has been given even though the spindle, lead axis or plane composing axes are controlled by drive other than MDS-E Series.
- The spindle intended for thread cutting is executing synchronous tapping, spindle synchronization, guide bushing spindle synchronization or spindle/C axis control.

Remedy

- Check the specifications.
- Check the parameters.
- Check the program.
- Check the Encoder selection signal (R register).

M01	Synchronization mismatch	1030

Details

Different M codes were each commanded as synchronization M code in each of the two part systems.
Synchronization with the "!" code was commanded in another part system during M code synchronization.
Synchronization with the M code was commanded in another part system during synchronization with the "!" code.

Remedy

- Correct the program so that the M codes match.
- Correct the program so that the same synchronization codes are commanded.

M01	Multiple C axes select invalid	1031

Details

The "C axis selection" signal has been changed when the multiple C axes selection is not available.
The selected axis by the "C axis selection" signal cannot be controlled for the multiple C axes selection.

Remedy

- Correct the parameter settings and program.

M01	Tap retract Sp select illegal	1032

Details

Tap retract has been executed with a different spindle selected. Cutting feed is in wait state until synchronization is completed.

Remedy

- Select the spindle for which tap cycle was halted before turning ON the "tap retract" signal.

M01	Sp-Sp polygon cut interlock	1033

Details

Cutting feed is in wait state until synchronization is completed.

Remedy

- Wait for the synchronization to end.

M01	Mixed sync ctrl prmtr illegal	1034

Details

There is a mistake in the settings of mixed control axis parameters (crsax [1] to [8]).
Mixed control was attempted within one and the same part system.
Any of the parameter settings is disabling mixed control.

Remedy

- Check the parameter settings for mixed synchronization control.

M01	Mixed sync ctrl disable modal	1035

Details

Mixed synchronization control was commanded for a part system in which mixed synchronization control is disabled as shown below.

- During nose R compensation mode
- During polar coordinate interpolation mode
- During cylindrical interpolation mode
- During balance cut mode
- During fixed cycle machining mode
- During facing turret mirror image
- During constant surface speed control mode
- During hobbing mode
- During axis name switch
- During interference check III alarm (interference detection, interference alarm area intrusion or interference warning area intrusion)

Mixed control was attempted even though the number of axes in the destination part system would exceed the maximum number when an axis control was shifted to another part system.

Mixed control was attempted even though the number of axes in the source part system would be "0" when an axis control was shifted to another part system.

Another axis exchange was attempted to the axis which was already transferred to another part system for mixed control.

Mixed control was attempted with an axis of a part system not in automatic operation.

Remedy

- Correct the program.

M01	Synchro ctrl setting disable	1036

Details

- The synchronous control operation method selection (R2589 register) was set while the C axis mode was inactive.
- The synchronous control operation method selection (R2589 register) was set while the zero point was undetermined.
- Mirror image disable state
- The external mirror image or parameter mirror image was commanded during facing turret mirror image.
- Synchronous control was started while either the master or slave axis is in the PLC axis mode due to NC axis/PLC axis switchover.
- The synchronous control mode was switched while the PLC axis mode was active due to NC axis/PLC axis switchover.
- Manual operation was attempted while the setting of the parameter "#2703 manual_acc" (Enable constant-gradient acceleration/deceleration in manual feed) is different between the master and slave axes.
- Synchronous control operation method (R2589) has been set to the axis in chopping operation.

Remedy

- Set the contents of the R2589 register to "0".
- Check the program and parameters.

M01	Synchro start/cancel disable	1037

Details

Synchronous control start/cancel command was issued when the start/cancel is disabled.

Remedy

- Correct the program and parameters.

M01	Move cmnd invld to synchro ax	1038

Details

A travel command was issued to a synchronized axis in synchronous control.

Remedy

- Correct the program.

M01	No spindle speed clamp	1043

Details

- When "#1146 Sclamp" has been set to "1", the constant surface speed command (G96) was issued to the spindle which is not selected for the spindle speed clamp command (G92/G50) under Multiple spindle control II.
- When "#1146 Sclamp" has been set to "0", both the G96 (Constant surface speed control ON) modal and the spindle forward or reverse rotation signal have turned ON for the spindle, on which the speed clamp command is disabled.

Remedy

Press the reset key and carry out the remedy below.

- Issue the G92/G50 command to the spindle that is to be used for the constant surface speed control.

M01	Cont ax superimpos II prm illg	1044

Details

- There is a mistake in the setting of the superimposition control reference axis parameter (#2089 bsax_pl).
- Superimposition control is not available under the current parameter settings.

Remedy

- Correct the parameter.

M01	Sync error btwn part systems	1045

Details

After a single block stop or automatic operation pause was executed during single block between part systems, cycle start has not been performed on either part system.

Remedy

Perform cycle start for all the part systems where a single block stop or automatic operation pause was executed.

M01	Hypothetical axis cmd invalid	1046

Details

The movement of the hypothetical axis (#1604 = 1) occurred due to the following reasons:

- A travel command was given to the hypothetical axis.
- A command that moves an axis other than the hypothetical axis caused the hypothetical axis to move.
- A tool compensation operation after a T command caused the hypothetical axis to move.

Remedy

- Check the machining program and sequence program to ensure that the hypothetical axis does not move.
- Set the tool compensation value applied to the hypothetical axis to zero.

M01	Z detect speed parameter error	1049

Details

Phase-Z detection speed has not been set in "#3109 zdetspd".

Remedy

- Set the phase-Z detection speed in "#3109 zdetspd".

M01	Intrf chck across sys: Set err	1050

Details

Setting of Interference check across part systems is incorrect.

Remedy

- Correct the setting of Interference check across part systems.

M01	Intrf chck across sys: Area AL	1051

Details

Interference is already occurring on the set interfering object.

Remedy

- Turn OFF the interference check valid signal, moving the axis to a position where the interfering objects do not collide each other, and then restart Interference check across part systems.

M01	Intrf check across sys: Alarm	1052

Details

A command has been given that causes the interfering objects to collide each other.

Remedy

- Correct the program.

M01	Interfe chk 0pt return incomp	1053

Details

Interference check between part systems had started while zero point return of the part system set interfering object has not completed.

Remedy

- Turn OFF the interference check valid signal, and complete zero point return in the all part systems set interfering object.

M01	Hob cut in interlock state	1056

Details

You attempted to execute a cutting feed block during speed- or phase-synchronization in hob machining.

Remedy

- The error is canceled when hob synchronization is completed. Wait until the synchronization completion.

M01	Feed-forward control disabled	1060

Details

Feed-forward control was attempted while any of the following functions was enabled.

- Control axis superimposition I/II
- Control axis synchronization between part systems I/II

Remedy

- Turn OFF the feed-forward control request signal.

M01	Invalid axis name selection No	1062

Details

An out-of-range value is set for the display axis name selection number.

Remedy

- Correct the sequence program.

M01	SpindleModRtAxCtrl not allowed	1063

Details

Commanded Spindle-mode Rotary Axis Control on a linear type rotary axis.

Remedy

- Turn the Spindle-mode Rotary Axis Control function OFF to clear the error, and then change the parameter "#8213 Rotation axis type" to rotary type. ("0" or "1")

M01	Superimposition axis param err	1070

Details

- Cutting feed clamp rate is unspecified for a superimposition-related axis. Or the specified rate is outside the setting range.
  "#2091 plclamp", "#2629 pl3clamp" or "#2630 pl3clamp2"
- Rapid traverse rate is unspecified for a superimposition-related axis.
  Or the specified rate is outside the setting range.
  "#2090 plrapid", "#2621 plrapid2",
  "#2626 pl3rapid", "#2627 pl3rapid2" or "#2628 pl3rapid3"

Remedy

- Set the cutting feed clamp rate of the superimposition-related axis within the setting range.
- Set the rapid traverse rate of the superimposition-related axis within the setting range.

M01	Tool retract & return disabled	1080

Details

Tool retract and return command has been given in any of the following modes:

- Mixed control (Cross axis control) I
- Mixed control (Cross axis control) II
- Control axis synchronization between part systems I
- Control axis synchronization between part systems II
- Control axis superimposition I
- Control axis superimposition II
- Arbitrary axis superimposition control
- Tool spindle synchronization IA (spindle - spindle polygon)
- Tool spindle synchronization IB (spindle - spindle polygon)
- Tool spindle synchronization IC (spindle - NC axis polygon)
- Tool spindle synchronization II (hob)
- Spindle superimposition control

Remedy

Turn OFF the transit point designation signal.

M01	Eccentric turn cmd invalid	1093

Details

- The Eccentric turning command was issued when the spindle corresponding to the rotary axis specified by the parameter "#8046 Ecc_rotax" is not in C-axis mode.
- The Eccentric turning command was issued in a mode where the Eccentric turning command cannot be issued.

  Control axis superimposition

  Arbitrary axis superimposition

  Control axis synchronization between part systems

- Manual arbitrary reverse run mode
- Program format switch is being requested
- The target axes of the Eccentric turning command are the synchronous axes.
- The number of rotary axes configured for Eccentric turning is either zero or more than one in the part system.

Remedy

- Remove the condition that causes the error.

M01	Overspeed in Eccentric turning	1094

Details

During Eccentric turning, one or more axes have exceeded the allowable speed limit.

Remedy

- Ensure that the speed of each axis during Eccentric turning does not exceed the value of "#2002 clamp" (or "#2110 Clamp(H-precision)" in the high-accuracy control mode).

M01	Not rest arb ax exch ax config	1100

Details

When the axis exchange status by arbitrary axis exchange control is set to be retained at power off, the axis state before power off cannot be restored.

Remedy

- Turn the power off and on again.

M01	Arbitrary axis unexchangeable	1101

Details

The axis declared in the arbitrary axis exchange command is incapable of being exchanged.

Remedy

- Check both the program of the part system that issues the arbitrary axis exchange command and that of the part system subject to the axis exchange. Correct the program blocks in which the command is issued or the axis exchange is executed, if necessary.

M01	Cross control axis exists	1102

Details

A manual travel command has been given to the axis being exchanged when manual interruption is disabled under cross machining control (when "#1435 crsman" = 0).

Remedy

- This error can be cancelled by either one of the following operations.
  1: Cancel the manual travel command
  2: Reset the NC

M01	Arbitrary ax superimp. sys err	1103

Details

- Arbitrary axis superimposition command has been issued in any part system other than the one that contains either the reference or superimposed axis of arbitrary axis superimposition control.
- Arbitrary axis superimposition cancel command has been issued in any part system other than the one that contains the superimposed axis of arbitrary axis superimposition control.

Remedy

- Correct the program so that the arbitrary axis superimposition command is given in an appropriate part system.

M01	Spindle speed fluctuation	1105

Details

The actual spindle speed has fluctuated exceeding the allowable range, while spindle speed fluctuation detection (G162) is active.

The number following "S", which is output together with this alarm, indicates the spindle where speed fluctuations were detected.

Remedy

- Unnecessarily large load may be applied to the spindle during automatic operation.
  Reduce the spindle load.
- To prevent this alarm from being output during spindle speed fluctuation detection (G162), set the parameter #1242 set14 BIT2 to ON.

M01	Sp synchro phase calc illegal	1106

Details

Spindle synchronization phase alignment command was issued while the "phase shift calculation request" signal was ON.

Remedy

- Correct the program.
- Correct the sequence program.

M01	Illegal cmd in SP oscillation	1108

Details

A function that cannot be used with spindle oscillation was commanded during spindle oscillation.

Remedy

- Command the function after finishing spindle oscillation.

M01	SP oscillation cmd illegal	1109

Details

Spindle oscillation was commanded during executing a function that cannot be used with spindle oscillation.

Remedy

- Command spindle oscillation after finishing the function that cannot used with spindle oscillation.

M01	SP oscillation set val illegal	1110

Details

Spindle oscillation was commanded while a value out of range is set as spindle oscillation amplitude or spindle oscillation frequency.

Remedy

- Check the setting values of the amplitude and frequency.

M01	Sub part system I call error	1111

Details

- Sub part system control I command (G122) has been given to a part system where the sub part system I operation mode is deactivated.
- Sub part system control I command (G122) has been given to a part system which is not reserved as the sub part system in M80.

Remedy

- Activate the sub part system I operation mode for the sub part system before issuing G122.
  Part systems marked "SUB" on the monitor screen are under the sub partsystem I operation mode.
- Command the sub part system start to the sub part system which is reserved with "#1483 SBS1_sys num" for M80.

M01	Sub part system II start error	1112

Details

When the sub part system control II has been commanded, no part system is left capable of being activated as a sub part system.

Remedy

- Do not exceed the maximum number of simultaneously activatable sub part systems when commanding G144.
- Set the parameter #1437 SBS2_Spec BIT0 to 0 if you wish to wait until the sub part system becomes capable of being activated.

M01	Constant surface speed rdndnt	1113

Details

- Constant surface speed is commanded from other part system to the spindle that is in the thread/ thread cycle or the tapping cycle/ synchronous tapping cycle.
- To the spindle in constant surface speed control, the thread/ thread cycle or the tapping cycle/ synchronous tapping cycle are commanded from other part system.

Remedy

- Check the program.

M01	Constant torque disabled	1114

Details

- Constant torque control is commanded to the axis which the parameter "#2296 SV096(TQC)" (Constant torque control: Stopper-direction torque) setting is "0".
- Constant or proportional torque stopper control is commanded to the axis which is in movement by automatic or manual operation.
- Constant torque control is canceled to the constant torque control axis in movement by automatic or manual operation.
- Constant torque control is commanded to the proportional torque stopper control axis.
- Constant torque control is commanded again during the axis movement by constant torque control cancel.
- Constant torque control axis is at stroke limit or H/W stroke end.

Remedy

- Check the program.
- Check the sequence program.

M01	P torque stopper disabled	1115

Details

- Proportional torque stopper control is commanded to the axis which the parameter "#2296 SV096(TQC)" (Constant torque control: Stopper-direction torque) setting is "0".
- Proportional torque stopper control is commanded to the axis which is in movement by automatic or manual operation.
- The axis movement is commanded to the axis which is in the proportional torque constant control.
- Proportional torque constant control is commanded again during the axis movement by proportional torque constant control cancel.
- Proportional torque constant control axis is at stroke limit or H/W stroke end.

Remedy

- Check the program.
- Check the sequence program.

M01	Droop cancel disabled	1116

Details

- Constant torque control droop cancel is commanded to the axis which is not in the constant/ proportional torque control.
- Constant torque control droop cancel is commanded to the axis in movement by automatic or manual operation.

Remedy

- Check the program.
- Check the sequence program.

M01	Cmnd disabled in droop cancel	1117

Details

The axis movement by automatic or manual operation is commanded to the axis which the constant torque control droop is being canceled.

Remedy

- Check the program.
- Check the sequence program.

M01	SP equivalent load factor over	1120

Details

The equivalent load factor of the spindle motor has exceeded the set threshold that causes the excess warning.

Remedy

- Change the operation pattern in a way that decreases the spindle motor load.

M01	Differential tap cmnd disabled	1131

Details

- Differential speed tap command has been given although any rotation command has not been input to the reference spindle that is under spindle superimposition control.
- Synchronous tap command has been given to the reference spindle that is under spindle superimposition control.

Remedy

- Cancel the synchronous tapping cycle.

M01	Spd clamp in differential tap	1132

Details

- A tap cycle or synchronous tap cycle command given to the superimposed spindle has caused the spindle's actual rotation speed to exceed the spindle clamp speed.

Remedy

- Correct the spindle rotation speed in synchronous tap cycle.

M01	Constant surface spd disabled	1133

Details

- A constant surface speed control command has been given to the reference or superimposed spindle during differential speed tapping under spindle superimposition control.
- A differential speed tapping command has been issued while constant surface speed control is executed on the reference or superimposed spindle that is under spindle superimposition control.

Remedy

- Cancel the synchronous tap cycle or constant surface speed control.

M01	Spindle sync cancel error	1135

Details

- Spindle synchronization cancel command has been issued during rotation of C axis under spindle sync C axis control.
- G113 (without the address H or D) has been issued during the spindle synchronization for two or more sync sets.

Remedy

- This operation error is cancelled when C axis stops. Issue a spindle sync cancel command after C axis has stopped.
- Use G113D or G113H to cancel the spindle synchronization.

M01	GB SP sync:Cancel sgnl illegal	1137

Details

The guide bushing spindle synchronization temporary cancel signal was turned ON/OFF when the master spindle and the guide bushing spindle were in one of the following modes.

- During rotation (when not stopped)
- During tap cycle synchronization mode
- During spindle synchronization mode
- During tool-spindle synchronization I (polygon machining) mode
- During tool-spindle synchronization II (hobbing) mode
- During spindle C axis control C axis mode
- During orientation/indexing

Remedy

- Check the ladder program.

M01	GB SP sync runs independently	1138

Details

- The reference spindle was commanded as a spindle related to tool spindle synchronization IC (polygon).
- The guide bushing spindle was commanded as a synchronous tapping spindle.
- The guide bushing spindle was commanded as a spindle related to spindle synchronization/tool spindle synchronization I (polygon)/tool spindle synchronization II (hobbing).

Remedy

- Check the program.

M01	Cmds illegal in spindle synchr	1139

Details

Synchronous tapping was issued for the reference spindle or the synchronized spindle during spindle synchronization.

Remedy

- Cancel the synchronous tapping cycle by reset.

M01	SP sync acc/dec time red	1141

Details

Although "Spindle synchronization: acceleration/deceleration time reduction valid" is enabled ("#3130 syn_spec/bit8" = 1), the connected drive unit doesn't support the function.

Remedy

- Check the parameter.

M01	Operation non-continuable(STO)	1151

Details

An axis in STO state exists in the part system.

Remedy

- Release the STO state and reset the NC.

M01	Operation non-continuable(SOS)	1152

Details

An axis in SOS state exists in the part system.

Remedy

- Release the SOS state and reset the NC.

M01	Same tool number being used	1160

Details

- The same spindle tool number is already being used.
- The specified tool cannot be shared by more than one spindle axis.

Remedy

- Do not share a spindle tool number being used.

M01	Prog check: work posn error	1215

Details

When the NC reset signal is input with the High-speed simple program check: Coordinate position check ON signal (Y76B) set to ON, the workpiece coordinate position is different from the position at the program start.

Remedy

- Correct the machining program.

M01	Prog check: machine posn error	1216

Details

When the NC reset signal is input with the High-speed simple program check: Coordinate position check ON signal (Y76B) set to ON, the machine coordinate position is different from the position at the program start.

Remedy

- Correct the machining program.

M01	NC/PLC axis switch illegal	1250

Details

The following operation was performed to an axis which can be switched over between NC axis and PLC axis.

- PLC axis switchover signal was turned ON or OFF when it was prohibited to switch over the axis.

Remedy

- Make sure the axis switchover status signal is OFF and change the ON/OFF of the axis switchover signal.

M01	No spec: Multi axis synch ctrl	1254

Details

You have set the R register for selecting synchronous control operation method, although the option of multiple axis synchronization control is OFF.

Remedy

- Set 0 in R2589 (Synchronous control operation method selection).

M01	Multiple secondary ax selected	1255

Details

For the multiple axis synchronization control, you selected two or more slave axes without selecting any master axis.

Remedy

- Correct the setting of R2589 (Synchronous control operation method selection).

M01	NC axis not switchable	1260

Details

- The trigger for NC axis switchover was activated for an axis not in a state where NC axis switchover is permissible.
- The trigger for NC axis switchover was activated specifying a number outside the range for the setting of NC-axis switchover.

Remedy

- Cancel the error by turning off the signal for requesting NC axis switchover (AXCHGCMD/bit8) for the corresponding axis, and modify the state of the axis before turning on the signal again.
- Correct the number for the setting of NC-axis switchover (AXCHGCMD/LO 8bit), and turn on the signal for requesting NC axis switchover (AXCHGCMD/bit8) again.

M01	Spindle not switchable	1261

Details

- The trigger for spindle switchover was activated for a spindle not in a state where spindle switchover is permissible.
- The trigger for spindle switchover was activated specifying a number outside the range for the setting of spindle switchover.

Remedy

- Cancel the error by turning off the signal for requesting spindle switchover (SPCHGCMD/bit8) for the corresponding spindle, and modify the state of the spindle before turning on the signal again.
- Correct the number for the setting of spindle switchover (SPCHGCMD/LO 8bit), and turn on the signal for requesting spindle switchover (SPCHGCMD/bit8) again.

M01	Sync ctrl op method chg dsbl	1270

Details

Synchronous control operation method (R2589) has been changed for the axis in which Synchronous control and chopping are used together.

Remedy

- Change the axis bits of Synchronous control operation method (R2589) back to the previous settings.

M01	Independent op ax chop invld	1271

Details

Chopping has been started up for the axis in Independent operation.

Remedy

- Reset or turn OFF "Chopping" signal (CHPS).

M01	Slave ax set as chopping ax	1272

Details

Chopping has been started up for the slave axis in Synchronous control operation.

Remedy

- Reset or turn OFF "Chopping" signal (CHPS).
- Specify the master axis as the chopping axis.

M01	Cmnd invalid during VCC mode	1300

Details

The issued command is disabled during the vibration cutting mode.

Remedy

- Correct the program.

M01	VCC mode command invalid	1301

Details

Vibration cutting mode start command has been issued although the command is disabled during the currently active mode.

Remedy

- Correct the program.

M01	VCC invalid	1302	Error cause number

Details

[When error cause No. is 0001]

Cutting is being performed without vibration cutting control.

[When error cause No. is 0002]

Vibration cutting is being performed with OMR-FF being disabled.

Remedy

[When error cause No. is 0001]

- Make sure that the "VCC: Temporary cancel of axis vibration" signal (R22532) is OFF.
- Make sure that the spindle speed command is not "0" when the cutting command is issued.
- Make sure that neither synchronized axis nor superimposed axis is included in the part system.

[When error cause No. is 0002]

- Make sure that the servo parameter "#2313 SV113" bit0 is "1" and bit1 is "0" for the VCC objective axes.

M01	Feedrate limited during VCC	1303

Details

The specified feedrate is being limited to the clamp feedrate during the vibration cutting mode.

Remedy

- Correct the program.

M01	VCC condition not found	1304

Details

No applicable condition is found during the selection of the vibration cutting condition.

Remedy

- Correct the program.
- Check the setting of the frequency upper limit "#12131 FrqClmpSys_VCC".
- Check the settings of the maximum spindle rotation speed "#3005 smax1" to "#3008 smax4" and the minimum rotation speed "#3023 smini".

M01	Cannot change SP speed in VCC	1307

Details

You changed the spindle rotation speed during vibration cutting control.

Remedy

- Wait for completion of vibration cutting control, or input Reset to cancel the VCC mode.

M01	Cannot re-run SP during VCC	1308

Details

You stopped the spindle during vibration cutting control, and then attempted to re-run it.

Remedy

- If you have stopped the spindle during vibration cutting control, input Reset to cancel the VCC mode before re-running the spindle.

M01	Vib. feedrate limited in VCC	1309

Details

The vibration feedrate is being limited during the vibration cutting mode.

Remedy

- Correct the program.
- Correct the setting of "#12586 ActualClamp_VCC" (Maximum vibration feedrate for each axis).

M01	VCC in sp speed limitation	1411

Details

Spindle rotation speed is limited so that vibration frequency does not exceed the Upper limit of frequency (#12131 FrqClmpSys_VCC).

Remedy

- Reduce the number of vibration.
- Reduce the spindle rotation speed.
- Check the setting value of Upper limit of frequency (#12131 FrqClmpSys_VCC).

M01	**Chatter supp set ax illegal**		1602

Details

- When chatter suppression was requested, the specified spindle did not exist.
- When chatter suppression was requested, a spindle that was under chatter suppression in another part system was specified.

Remedy

- Correct the setting.

M01	**No spec: Chatter suppression**		1607

Details

Chatter suppression is not included in the specifications.

Remedy

- Turn off the chatter suppression request signal.

M01	**Chatter suppression SP stop**		1608

Details

The spindle used in chatter suppression is stopped.

Remedy

- Rotate the spindle.
- Turn off the chatter suppression request signal.
- Check for a broken wire in the spindle encoder cable.
- Check the connections for the spindle encoder connectors.
- Check the spindle encoder pulse.

M01	**Orbit_CMND not allowed**		1609

Details

An attempt is made to run a command not allowed during the Orbit machining.

Remedy

- Correct the program.

M01	**HPU in preparation**		1620

Details

HPU custom module is being registered and deleted.

When this error is being displayed, HPU custom module cannot be input, restored or executed cycle start.

This alarm is exported to NC alarm 5.

Remedy

- When the registration and deletion of the HPU custom module is completed, an error will be canceled automatically.

M03	**Interference detection**	0001	yyzz

Details

A command has been given that causes two or more objects to interfere with each other.

yy : Interference object definition number (1)

zz : Interference object definition number (2)

Remedy

- Cancel the alarm with reset.
- Disable the interference check III mode first before moving the interfering object further in the interference direction.

M03	**Entry in interfere alarm area**	0002	yyzz

Details

Two or more interfering objects are in the interference alarm area.

yy : Interference object definition number (1)

zz : Interference object definition number (2)

Remedy

- Disable the interference check III mode before moving.
- Check the interference definition and the interference selection.

M03	**Entry in interfere warn area**	0003	yyzz

Details

A command has been given that causes two or more interfering objects to enter the interference warning area.

yy : Interference object definition number (1)

zz : Interference object definition number (2)

Remedy

- Interfering objects are in the interference warning area.

If an operational problem is expected to occur, stop moving the axes with feed hold, cancellation of manual axis movement or reset.

M03	**Interference data unspecified**	1001

Details

You attempted to turn ON the interference check III mode with the interfering object data disabled.

Remedy

- Execute the interference data ON command to enable the interference data before turning ON the interference check III mode.

M03	**Interf. 3D objects count error**	1002

Details

The total number of solid objects used as interfering object has exceeded the maximum.

Remedy

- The total number of interfering solid objects set in the system variable or R register has exceeded the maximum. Correct the setting.

M03	**Interfere check III: time over**	1003

Details

The allowable process time for interference check III has been exceeded.

Remedy

- Turn OFF the interference check III mode.
- Correct the interfering object definition.

M03	**Too many interfering objects**	1004

Details

The total number of selected interfering objects enabled by the interfering object enable/disable designation exceeds the maximum number.

Remedy

- The total number of interfering objects in interfering object selection specified with system variables or R registers exceeds the maximum number. Correct the settings.

M03	**Interfere: control axis error**	2001	Interference object definition number

Details

In the interfering object definition, the IJK control axes have been set as follows:

- Nonexistent axis name (an axis not set in the base axis specification parameter "#1022 axname2")
- I, J and K axes belong to different part systems
- Rotary axis

Remedy

- Correct the setting of the IJK control axes in the interfering object definition.

M03	**Interf:2 rot axes in cylinder**	2002	Interference object definition number

Details

In the interfering object definition, two or more rotary axes have been set for the cylindrical solid definition No.

Remedy

- Correct the solid designation and the setting of the IJK axis rotation angle in the interfering object definition.

M03	**Interfere:2 rot axes in cuboid**	2003	Interference object definition number

Details

In the interfering object definition, two or more rotary axes have been set for the cuboidal solid definition No.

Remedy

- Correct the setting of the IJK axis rotation angle in the interfering object definition.

M03	**Interfere: rotary axis error**	2004	Interference object definition number

Details

In the interfering object definition, the IJK rotary axes have been set as follows:

- Nonexistent axis name
  (an axis not set in the base axis specification parameter (#1022 axname2))
- Linear axis
- All three axes are rotary axes

Remedy

- Correct the setting of the IJK rotary axes in the interfering object definition.

M03	**Interfere:2 rot axes object er**	2005	Interference object definition number

Details

The solid object set in the interfering object definition is unavailable as an interfering object defined with two rotary axes.

Remedy

- Correct the setting of the solid in the interfering object definition.

M03	**Interfere:1 rot axis object er**	2006	Interference object definition number

Details

The solid object set in the interfering object definition is unavailable as an interfering object defined with one rotary axis.

Remedy

- Correct the setting of the solid in the interfering object definition.

M03	**No spec: int obj def warn area**	2007

Details

The interference warning area is set with interfering object definition or interfering object selection.

Remedy

- Correct the settings of the configured solid specification of interfering object definition or the interfering object specification of interfering object selection.

M03	**Too many int obj config solids**	2008

Details

The number of solids comprising an interfering object exceeds the maximum number.

Remedy

- Correct the solid designation in the interfering object definition.

M03	**Interf. selection: offset over**	3001	Interference object definition number

Details

In the interfering object selection, the interfering object model coordinate system offset 1 has exceeded the setting.

Remedy

- Correct the setting of the interfering object model coordinate system offset 1 in the interfering object selection.

M03	**Interf:rot objects comb. error**	3002	Interference object definition number

Details

The following selection was made for the rotary interfering objects:

- You selected a one-rotary-axis object and two-rotary-axis object at a time.
- You selected two or more two-rotary-axis objects.

Remedy

- Correct the setting of the n-th interfering object selection.

M50	**WCS offset not reflected**	5000

Details

Any of the following items being selected has not been reflected in the workpiece position counter or in the program position counter: workpiece coordinate system offset, extended workpiece coordinate system offset, external workpiece coordinate system offset, workpiece coordinate system shift or workpiece base point offset.

Remedy

Perform one of the following actions to cancel the warning.

- Cycle start
- Reset
- Emergency stop

M50	**Multiple key inputs invalid**	5002

Details

- Multiple key presses were detected.
  0xxx: indicates the first detected key-code.
- Simultaneous key presses were detected.
  No key-code is indicated.

Remedy

- Go to the alarm history to check the key-codes.
  Clean the keys and check the performance.
- Press one key at a time.

M50	**Continuous write of parameter**	5003

Details

Parameter is being written in every PLC scan cycle using a "write window" of the PLC window function. This may affect the performance.

Remedy

- Correct the user PLC (ladder sequence).

M50	**Spindle/C axis displacement**	5004

Details

A displacement occurred when switching axis from spindle mode to C axis mode.

Remedy

- Contact our service center.
- Cancel the alarm by resetting.

M50	**Cannot start multi-axis sync**	5005

Details

The bit of the register R2589 (Synchronous control operation method) that corresponds to the multiple-axis synchronization control axis has been set to "1" while multiple-axis synchronization control is disabled.

Remedy

- Set the bits of R2589 (Synchronous control operation method) that correspond to the multiple-axis synchronization control axes to "0".

M50	BrakeWearDiagnostics (Caution)	5006

Details

The axis travel amount exceeded the brake wear diagnostics allowable travel amount (Caution) during brake wear diagnostics.

The axis name of the axis where the error occurred is displayed.

Remedy

- Move the axis to a safe position with manual operation, turn off the power, and take corrective action for the brake. This warning is cleared upon successful completion of brake wear diagnostics, which is performed after the power is turned back on.
- This warning can be cleared by the brake wear diagnostics alarm reset signal. The brake wear diagnostics incomplete signal (BWDNFm) remains on.

M50	Brake wear diag param invalid	5007

Details

The brake wear diagnostics parameters are incorrectly configured.

- The value of "#12597 BWD_TOL(Caution)" is greater than or equal to "#12596 BWD_TOL(Warning)". The operation error (M50 5006) is not generated.

Remedy

- Correct the parameter settings so that "#12597 BWD_TOL(Caution)" value is smaller than "#12596 BWD_TOL(Warning)" value.

M50	Email notif to opr setting chg	5008

Details

The e-mail address or the server setting to be used for "Email notification to operator" function has been changed.

Remedy

- If this alarm occurred unintentionally, check the email address, user ID, and password settings.
- Cancel the alarm by resetting.

M90	Parameter set mode	

Details

The setup parameter lock is released.
Cycle start is disabled when setup parameters can be set.

Remedy

- Refer to the manual issued by the machine tool builder.

M91	TLM error	0002

Details

Data is out of range:

- The result of the measurements is out of the range of the tool data.

Remedy

- Check the value of "#2015 tlml-" and "#2016 tlml+".

M91	TLM error	0003

Details

No valid number found:

- The tool number to perform the measurement has not been specified.
- Or the specified tool number is out of range.

Remedy

- Check that the tool number to perform the measurement is correct.

M91	TLM error	0045

Details

Illegal axes measured:

- The sensor is pressed while more than one axis is moving.

Remedy

- Only one axis can be moved to press the sensor.

M91	TLM error	0046

Details

Reference position return not performed:

- The measured axis has not yet performed reference position return when using incremental coordinates.

Remedy

- Perform the measurement after the axis to measure performs reference position return.

M91	TLM error	0089

Details

Illegal sensor input without axis feed:

- The sensor input has already been detected when the TLM mode is enabled.
- The axis feed may be insufficient when the sensor is pressed.

Remedy

- All axes are interlocked by this error.
  Set the TLM signal to "OFF" or clear the interlock signal(s) to move the axis away from the sensor.
  The distance to move the axis toward the sensor must be 0.1 mm or greater.

M91	TLM error	9000

Details

Lower than minimum speed:

- The tool is pressed into the sensor at a feed rate lower than "#1508 TLM_Fmin".

Remedy

- Verify the feed rate towards the sensor.

M91	TLM error	9001

Details

Higher than maximum speed:

- The tool is pressed into the sensor at a feed rate higher than "#1509 TLM_Fmax".

Remedy

- Verify the feed rate towards the sensor.

M91	TLM error	9002

Details

Signal state updated illegally:

- Compensation number and/or "Tool pre-setter sub-side valid" signal has been illegally updated while the sensor signal is "ON" or the compensation data is being written to the device.

Remedy

- Retry the measurement.

M91	TLM error	9003

Details

Untimely sensor response received:

- The compensation number was updated at the same time the sensor response was received.

Remedy

- Retry the measurement.

M92	TLM: Interlock not working

Details

Manual Tool Length Measurement-Interlock temporary NOT working:

- The operation errors ("M01 Internal interlock axis exists 0005", "M01 Sensor signal illegal ON 0019") generated by the Manual Tool Length Measurement are temporary invalidated.
- This message is necessary to indicate that the interlock is not working while the axis is being moved to the safe position when the interlock is temporarily invalidated due to the error condition such as M01 0019.

Remedy

- After the tool is moved away from the sensor, set "Manual tool length measurement interlock temporarily canceled" signal to "OFF".

M95	**Insuf lsr proc cond in DR ctrl**	9501

Details

Only one or no laser processing condition has been set for DR control.

Remedy

- Set two or more laser processing conditions with different speeds for DR control.

M95	**DR ctrl invalid (Laser power)**	9502

Details

The laser power command by DR control is less than or equal to 0.

Remedy

- Correct the laser processing condition for DR control.

M95	**DR ctrl invalid (Frequency)**	9503

Details

The frequency command by DR control is less than or equal to 0.

Remedy

- Correct the laser processing condition for DR control.

M95	**DR ctrl invalid (Duty)**	9504

Details

The duty command by DR control is less than or equal to 0.

Remedy

- Correct the laser processing condition for DR control.

M95	**Excess max stop time w beam ON**	9505

Details

There has been no axis movement with laser being ON for a longer period than specified with "#90005 lsr_-max_stop_t" (Maximum stop time with beam ON).

Remedy

- Correct the machining program.

M95	**Laser prcsg initial set error**	9506

Details

The parameters required for enabling laser processing control have not been configured.

Remedy

Correct the parameters.

- "#1026 base_I" (Base axis I) to "#1028 base_K" (Base axis K)
- "#8090 SSS ON" = 1
- "#12066 Tolerance ctrl ON" = 1

M95	**Laser prcsg parameter error**	9507

Details

A function that cannot be used in the part system where laser processing is active has been enabled.

Remedy

Correct the parameter.

- "#12060 VblAccPreInt" (Variable-acceleration pre-interpolation acceleration/deceleration control while high-accuracy control) = 0
- "#1190 s_xcnt" (Inclined-axis control) = 0

M95	Beam ON impossible	9508

Details

Laser beam was turned on when beam ON was not permissible.

- The "Laser beam irradiation ready" signal is not on.
- Miscellaneous function strobe 1 to 4 are ON.
- The laser condition change strobe (X1CA6) is ON.
- The laser processing function is not included in the specifications.

Remedy

- Perform the preparatory operation required for beam ON, and turn on the "Laser beam irradiation ready" signal.
- When the laser processing condition is switched using an M code, turn on the "M function finish" signal.
- When the current laser processing condition was changed using the laser processing condition selection screen or the laser processing condition setting screen, turn on the "Laser condition change complete" (Y1CA6).
- Check the specifications.

M95	Ax stop time w beam ON mon OFF	9509

Details

Monitoring of axis stop time with beam ON has been disabled.

Remedy

- After power calibration has completed, turn off the "Axis stop time with beam ON monitoring OFF" signal.

M95	Hgt ctrl ax travel impossible	9510

Details

- An attempt was made to execute a travel command for the height control axis during height control.
- An attempt was made to move the height control axis manually during height control.

Remedy

- Correct the program.
- Turn off height control before moving the height control axis.

M95	Power calibration data invalid	9511

Details

The power calibration data contains a value outside the allowable range.

Remedy

- Correct the power calibration data in "#90121 lsr_calib_cmd01" through "#90180 lsr_calib_meas30".

M95	Fast retract ctrl: Inv retract	9512

Details

The specified retraction amount for fast retraction control exceeds the parameter "#90107 lsr_HEvac_rise_lim" (Height control_Fast retraction control lift upper limit).

Remedy

- Correct address R for fast retraction control or the parameter "#90107 lsr_HEvac_rise_lim" (Height control_-Fast retraction control lift upper limit).

M97	Maintenance part activated

Details

Activated maintenance part has not completed the product procedures.

Remedy

- Contact our service center.

2

Stop Codes (T)

T01	Axis in motion	0101

Details

Automatic start is not possible as one of the axes is moving.

Remedy

- Try automatic start again after all axes have stopped.

T01	NC not ready	0102

Details

Automatic start is not possible as the NC is not ready.

Remedy

- Another alarm has occurred. Check the details and remedy.

T01	Reset signal ON	0103

Details

Automatic start is not possible as the "reset" signal has been input.

Remedy

- Turn OFF the "reset" signal.
- Check for any failure of the reset switch which has caused the switch's continuous ON.
- Correct the sequence program.

T01	Auto operation pause signal ON	0104

Details

The feed hold switch on the machine operation panel is ON (valid).

Remedy

- Correct the feed hold switch setting.
- The feed hold switch is B contact switch.
- Fix any broken wires in the feed hold signal line.
- Correct the sequence program.

T01	H/W stroke end axis exists	0105

Details

Automatic start is not possible as one of the axes is at the stroke end.

Remedy

- Manually move any axis whose end is at the stroke end.
- Check for any broken wires in the stroke end signal line.
- Check for any failure in the stroke end limit switch.

T01	S/W stroke end axis exists	0106

Details

Automatic start is not possible as one of the axes is at the stored stroke limit.

Remedy

- Move the axis manually.
- If the axis's end is not at the stroke end, check the parameters.

T01	No operation mode	0107

Details

The operation mode has not been selected.

Remedy

- Select automatic operation mode.
- Check for any broken wires in the signal line for automatic operation mode (memory, tape, MDI).

T01	Operation mode duplicated	0108

Details

Two or more automatic operation modes have been selected.

Remedy

- Check for any short circuit in the mode (memory, tape, MDI) selection signal line.
- Check for any failure in the switch.
- Correct the sequence program.

T01	Operation mode changed	0109

Details

The automatic operation mode has changed to another automatic operation mode.

Remedy

- Return to the original automatic operation mode, and execute automatic start.

T01	Tape search execution	0110

Details

Automatic start is not possible as tape search is being executed.

Remedy

- Wait for the tape search to be completed and then execute the automatic start.

T01	Cycle start prohibit	0111

Details

Automatic start is disabled because restart search is in execution.

Remedy

- Execute automatic start after the restart search is completed.

T01	CNC overheat	0113

Details

Automatic start is not possible because a thermal alarm (Z53 CNC overheat) has occurred.

Remedy

- Temperature of the control unit has exceeded the specified temperature.
- Take appropriate measures to cool the unit.

T01	Cycle st. prohibit(Host comm.)	0115

Details

Automatic start cannot is not possible because the NC is communicating with the host computer.

Remedy

- Wait for the communication with host computer to be ended and then execute the automatic start.

T01	Cycle st prohibit(Battery alm)	0116

Details

Automatic start is not possible because the voltage of the battery in the NC control unit has dropped.

Remedy

- Replace the battery of the NC control unit.
- Contact the service center.

T01	R-pnt offset value not set	0117

Details

Automatic operation is not possible because no reference position offset value has been set.

Remedy

- Perform the reference position initialization setting, then set "#2034 rfpofs(Distance-coded reference position detection offset)".

T01	In absolute position alarm	0138

Details

♦A start signal was input during an absolute position detection alarm.

Remedy

♦Clear the absolute position detection alarm, and then input the start signal.

T01	In abs posn initial setting	0139

Details

♦A start signal was input during zero point initialization in the absolute position detection system.

Remedy

♦Complete zero point initialization before inputting the start signal.

T01	In manual measurement	0143

Details

Automatic start is disabled because manual measurement is in execution.

Remedy

♦Execute automatic start after the manual measurement is completed.

T01	Sub part sys I mode is active	0146

Details

Cycle start signal was input for the part system that has applied Sub-part system I operation mode.

Remedy

♦Use Sub-part system I operation mode signal to switch whether to start the operation as Sub-part system control or to execute cycle start as Main-part system.

T01	Mach. interrupt non-startable	0147

Details

Machining interruption related functions cannot be started up.

Remedy

♦To resume the operation from the state of being stopped at the selected point, set the automatic operation mode to the memory mode.
♦To perform the retraction, confirm that retraction is enabled (the "Retraction executable" signal is ON).
♦To resume the operation from the state of being stopped at a certain block of the retraction program, activate the memory mode and turn ON the "Retraction start" signal.
♦Check the sequence program to make sure that multiple start signals are not input at the same time.

(Note) The remedies stated above apply not only to retraction but also to the other machining interruption-related functions. (The start signals and ready states are replaced with those for each function.)

T01	APLC password mismatch	0160

Details

Automatic start is disabled because the APLC authentication password is inconsistent.

Remedy

♦Contact the machine tool builder.

T01	Cycle st. prohibit(safe speed)	0165

Details

An automatic operation was started in one of the following machining modes during the safe speed clamp: thread cutting, synchronous tapping, cross machining, superimposition control, tool center point control or SSS control.

Remedy

♦Turn ON the safely-limited speed monitoring request signal and press the cycle start button.
♦Close the door and press the cycle start button.

T01	Cannot start Cyc OP (BWD WARN)	0166

Details

Cycle operation cannot be started because of the operation error "M01 0371 BrakeWearDiagnostics (Warning)".

Remedy

- Clear the operation error (M01 0371), and then start cycle operation.

T01	Cannot start Cyc OP (in BWD)	0167

Details

Cycle operation cannot be started because there is an axis under brake wear diagnostics in the system.

Remedy

- After the completion of brake wear diagnostics, start cycle operation.

T01	Cycle start prohibit (AL Z83)	0172

Details

Cycle start is disabled while the system alarm Z83 (NC started during SP rotation) is occurring.

Remedy

- Turn OFF the power and make sure that the spindle is at a standstill. And then turn the power ON again.

T01	In interference check alarm	0173

Details

Cycle start cannot be operated because interference check alarm is in progress.

Remedy

- Release the interference check alarm before operating the cycle start.

T01	Cycle st forbid (measuring err)	0176

Details

Cycle start cannot be executed for other part systems during measurement of rotation center error or workpiece installation error.

Remedy

Execute cycle start for one part system that is ready for measurement.

Cancel measurement when error measurement is not executed.

T01	Cycle start prohibit (OP error)	0177

Details

Cycle start was executed during operation error where cycle start cannot be executed.

Remedy

After canceling operation error, execute cycle start.

T01	Cycle start prohibit(SBT warn)	0185

Details

Cycle start is disabled because the axis in SBT warning exists in the system.

Remedy

After cancelling the SBT warning, execute the cycle start.

T01	Cycle start prohibit	0190

Details

Automatic start is not possible because the setting of setup parameters is enabled.

Remedy

- Refer to the manual issued by the machine tool builder.

T01	Cycle start prohibit	0191

Details

Automatic start was attempted while a file was being deleted/written.

Remedy

- Wait for the file to be deleted/written and then execute the automatic start.

T01	Cycle st. prohibit (Term exp'd)	0193

Details

Automatic start is not possible because the valid term has been expired.

Remedy

- Obtain a decryption code from the machine tool builder and input it in the NC, then turn the power ON again.

T01	Cycle start disabled (in SBT)	0194

Details

Cycle start is disabled because the break test is being executed for some axes in the system.

Remedy

- Execute cycle start after the break test is completed.

T01	Cycle start prohibit (HD mode)	0196

Details

Cycle start is disabled because the HD operation option is OFF.

Remedy

- If you store user programs in the memory, set "#11053 UserProgramStorage" (User program storage destination) to "0" and then restart the NC.
- If you store user programs in the hard disk, add the HD operation option.

T01	Cycle start prohibit(in SBT)	0197

Details

Cycle start is disabled because the axis in brake test exists in the system.

Remedy

After the brake test completed, execute the cycle start.

T02	EMG stop	0200

Details

An emergency stop occurred during automatic operation.

Remedy

- After the emergency stop is canceled, the operation can be continued.

T02	H/W stroke end axis exists	0201

Details

An axis is at the stroke end.

Remedy

- Manually move the axis away from the stroke end limit switch.
- Correct the machining program.

T02	S/W stroke end axis exists	0202

Details

An axis is at the stored stroke limit.

Remedy

- Manually move the axis.
- Correct the machining program.

T02	Reset signal ON	0203

Details

The reset has been entered.

Remedy

- The program execution position has returned to the start of the program. Execute automatic operation from the start of the machining program.

T02	**Auto operation pause signal ON**	0204

Details

The "feed hold" switch is ON.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T02	**Operation mode changed**	0205

Details

The operation mode has changed to another mode during automatic operation.

Remedy

- Return to the original automatic operation mode, and press the CYCLE START switch to resume the automatic operation.

T02	**Acc/dec time cnst too large**	0206

Details

The acceleration and deceleration time constants are too large. (This alarm occurs with the system alarm Z59.)

Remedy

- Set a larger value for "#1206 G1bF(Maximum speed)".
- Set a smaller value for "#1207 G1btL(Time constant)".
- Set a lower cutting speed.

T02	**Abs posn detect alarm occurred**	0215

Details

An absolute position detection alarm occurred.

Remedy

- Clear the absolute position detection alarm.

T02	**Aux axis changeover error**	0220

Details

A travel command was issued to an auxiliary axis.

Remedy

- Turn ON the "NC axis control selection" signal and press the CYCLE START switch to restart the automatic operation with.

T02	**FEED HOLD: SOS axis**	0221

Details

SOS axis exists in the system.

Remedy

Cancel SOS state and execute cycle start, and you can continue automatic operation from the stop point.

T03	**Single block stop signal ON**	0301

Details

The SINGLE BLOCK switch on the machine operation panel is ON.
The SINGLE BLOCK or MACHINE LOCK switch changed.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03	**Block stop cmnd in user macro**	0302

Details

A block stop command was issued in the user macro program.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03	Operation mode changed	0303

Details

Automatic mode changed to another automatic mode.

Remedy

- Return to the original automatic operation mode, and press the CYCLE START switch to resume the automatic operation.

T03	MDI completed	0304

Details

MDI operation has ended the last block.

Remedy

- Set the MDI operation again, and press the CYCLE START switch to start the MDI operation.

T03	Block start interlock	0305

Details

The interlock signal, which locks the block start, is ON.

Remedy

- Correct the sequence program.

T03	Cutting blck start interlock	0306

Details

The interlock signal, which locks the block cutting start, is ON.

Remedy

- Correct the sequence program.

T03	Safe speed clamp is ON	0308

Details

One of the following commands was issued during the safe speed clamp:
thread cutting, synchronous tapping, cross machining, superimposition control, tool center point control or SSS control.

Remedy

- Turn ON the safely-limited speed monitoring request signal before performing the operation.
- Perform the operation with the door closed.

T03	Inclined Z offset change	0310

Details

The "inclined axis control: No Z axis compensation" signal has turned ON or OFF during the program operation.

Remedy

- Press the CYCLE START switch to resume the automatic operation.

T03	Stop at selected point	0320

Details

Operation has stopped at the selected point.

Remedy

- Perform the automatic start to execute the program from the block.

T03	Aux axis changeover error	0330

Details

The "NC axis control selection" signal was turned OFF while a NC axis was traveling.

Remedy

- Turn the "NC axis control selection" signal ON and press the CYCLE START switch to resume the automatic operation.

T04	Collation stop	0401

Details

Collation stop occurred.

Remedy

- Execute the automatic start to resume the automatic operation.

T10	Fin wait	(Factors for waiting completion)

Details

The following Nos. are shown during the operation of the corresponding completion wait factor. The numbers will disappear when the operation is completed.
The completion wait factor is indicated with four digits (in hexadecimal).

Display format of completion wait factor

0__ __ __
(a)(b)(c)

Each of the hexadecimal numbers (a), (b) and (c) indicates the following details.

(a)
bit0: In dwell execution
bit3: Unclamp signal wait (Note 1)

(b)
bit0: Waiting for spindle position to be looped
bit3: Door open (Note 2)

(c)
bit0: Waiting for MSTB completion
bit1: Waiting for rapid traverse deceleration
bit2: Waiting for cutting speed deceleration
bit3: Waiting for spindle orientation completion

(Note 1)

For type A (#1282 ext18/bit3 = 0)

- Waiting for turning ON or OFF of the unclamp finish signal for the index table indexing.

For type B (#1282 ext18/bit3 = 1)

- Waiting for turning ON of the unclamp finish signal for the index table indexing.
- Waiting for turning ON of the clamp finish signal for the index table indexing.

(*) The complete standby status is established when the unclamp finish and clamp finish signals are both turned ON. Set the clamp finish signal to OFF after performing required process with the PLC.

(Note 2) This shows the door open state caused by the door interlock function.

T11	Fin wait	(Factors for waiting completion)

Details

The following Nos. are shown during the operation of the corresponding completion wait factor. The numbers will disappear when the operation is completed.
The completion wait factor is indicated with four digits (in hexadecimal).

Display format of completion wait factor

0__ __ __
(a)(b)(c)

Each of the hexadecimal numbers (a), (b) and (c) indicates the following details.

(b)
bit0:Operation alarm display being postponed

Remedy

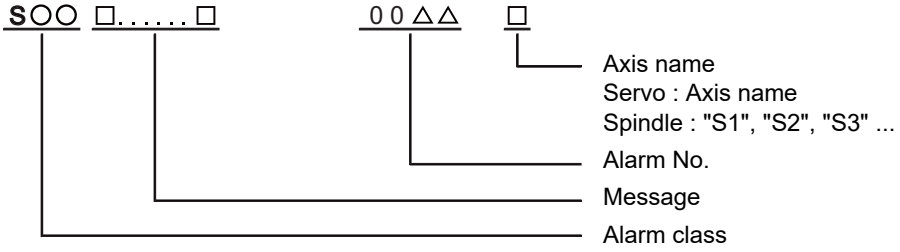
- The parameter "#1342 AlmDly" may be able to postpone displaying a part of an operation alarm, depending on the setting.
  This stop code will remain displayed while any alarm is being postponed.
  And it will disappear if the postponed alarm is displayed or canceled.

3

Servo/Spindle Alarms (S)

3.1 Servo Errors (S01/S03/S04)

Servo alarm is displayed in the following format.



Alarm class	Message	Reset method	Resetting methods
S01	Name or "Servo alarm"	PR	After removing the cause of the alarm, reset the alarm by turning the NC power ON again.
S03	Name or "Servo alarm"	NR	After removing the cause of the alarm, reset the alarm by inputting the NC RESET key.
S04	Name or "Servo alarm"	AR	After removing the cause of the alarm, reset the alarm by turning the NC and drive unit power ON again.

The numerical value in the column of "No." in the table after the following page is the double-digit number displayed on the LED which is on the front-side of the drive unit.

At the time of the occurrence of the servo errors (S01/S03/S04), this number will be displayed in the last two digits of the "alarm number".

(Note 1) For the troubleshooting at the time of the occurrence of the servo errors, refer to your drive unit's instruction manual.

(Note 2) When multiple servo alarms have occurred, only an alarm which occurred first is displayed on the alarm screen. Refer to the alarm history on the drive monitor screen as necessary.

(1) Drive unit alarm

No.	Name	Details	Reset method	Servo stop method	Spindle stop method
10	Insufficient voltage	A drop of bus voltage was detected in main circuit.	PR	Dynamic stop	Coast to a stop
11	Axis selection error	The axis selection rotary switch has been incorrectly set.	AR	Initial error	Initial error
12	Memory error 1	A hardware error was detected during the power ON self-check.	AR	Initial error	Initial error
13	Software processing error 1	An error was detected for the software execution state.	PR	Dynamic stop	Coast to a stop
14	Software processing error2	The current processor is not operating correctly.	AR	Dynamic stop	Coast to a stop
16	Initial magnetic pole position detection error	In the built-in motor which uses the absolute position encoder, the servo ON has been set before the magnetic pole shift amount is set. The magnetic pole position, detected in the initial magnetic pole position detection control, is not correctly set.	PR	Dynamic stop	Coast to a stop
17	A/D converter error	A current feedback error was detected.	PR	Dynamic stop	Coast to a stop
18	Main side encoder: Initial communication error	An error was detected in the initial communication with the motor side encoder.	PR	Initial error	Initial error
19	Encoder communication error in synchronous control	An error of the shared encoder on the machine side was detected on the secondary axis of the speed command synchronization control.	PR	Dynamic stop	-
1A	Sub side encoder: Initial communication error	An error was detected in the initial communication with the machine side encoder.	PR	Initial error	Initial error
1B	Sub side encoder: Error 1	An error was detected by the encoder connected to the machine side. The error details are different according to the encoder type. Refer to "Encoder alarm" for details.		Dynamic stop	Coast to a stop
1C	Sub side encoder: Error 2				
1D	Sub side encoder: Error 3				
1E	Sub side encoder: Error 4				
1F	Sub side encoder: Communication error	An error was detected in the communication with the machine side encoder.	PR	Dynamic stop	Coast to a stop
21	Sub side encoder no signal 2	In the machine side encoder, ABZ-phase feedback cannot be returned even when the motor moves.	PR	Dynamic stop	Coast to a stop
22	Encoder data error	An error was detected in the feedback data from the position encoder.	AR	Dynamic stop	Coast to a stop
23	Excessive speed error	The state that there is a difference between the actual speed and command speed continued for longer than the excessive speed deviation timer setting.	NR	-	Coast to a stop
24	Grounding	The motor power cable is in contact with FG (Frame Ground).	PR	Dynamic stop	Coast to a stop
25	Absolute position data lost	The absolute position data was lost in the encoder.	AR	Initial error	-
26	Unused axis error	In the multiaxis drive unit, there is an axis set to free, and the other axis detected a power module error.	PR	Dynamic stop	Coast to a stop

(Note 1) Definitions of terms in the table are as follows.

Main side encoder: Encoder connected to CN2

Sub side encoder: Encoder connected to CN3

(Note 2) Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions. When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the NC and servo drive unit power ON again.

No.	Name	Details	Reset method	Servo stop method	Spindle stop method
27	Sub side encoder: Error 5	An error was detected by the encoder connected to the machine side. The error details are different according to the encoder type. Refer to "Encoder alarm" for details.		Dynamic stop	Coast to a stop
28	Sub side encoder: Error 6				
29	Sub side encoder: Error 7				
2A	Sub side encoder: Error 8				
2B	Main side encoder: Error 1	An error was detected by the encoder connected to the motor side. The error details are different according to the encoder type. Refer to "Encoder alarm" for details.		Dynamic stop	Coast to a stop
2C	Main side encoder: Error 2				
2D	Main side encoder: Error 3				
2E	Main side encoder: Error 4				
2F	Main side encoder: Communication error	An error was detected in the communication with the motor side encoder.	PR	Dynamic stop	Coast to a stop
30	Over regeneration	Over-regeneration level exceeded 100%. The regenerative resistor is overloaded.	PR	Dynamic stop	Coast to a stop
31	Overspeed	The motor speed exceeded the allowable speed.	PR	Deceleration stop	Deceleration stop
32	Power module error (overcurrent)	The power module detected the overcurrent.	PR	Dynamic stop	Coast to a stop
33	Overvoltage	The bus voltage in main circuit exceeded the allowable value.	PR	Dynamic stop	Coast to a stop
34	NC communication: CRC error	The data received from the NC was outside the setting range.	PR	Deceleration stop	Deceleration stop
35	NC command error	The travel command data received from the NC was excessive.	PR	Deceleration stop	Deceleration stop
36	NC communication: Communication error	The communication with the NC was interrupted.	PR	Deceleration stop	Deceleration stop
37	Initial parameter error	An incorrect set value was detected among the parameters send from the NC at the power ON. In the SLS(Safely Limited Speed) function, an error was detected in the relation between the safety speed and safety rotation number in the speed observation mode.	PR	Initial error	Initial error
38	NC communication: Protocol error 1	An error was detected in the communication frames received from the NC. Or, removing an axis or changing an axis was performed in the synchronous control.	PR	Deceleration stop	Deceleration stop
39	NC communication: Protocol error 2	An error was detected in the axis data received from the NC. Or, in changing an axis, the parameter setting of the synchronous control was applied when the axis was installed.	PR	Deceleration stop	Deceleration stop
3A	Overcurrent	Excessive motor drive current was detected.	PR	Dynamic stop	Coast to a stop
3B	Power module error (overheat)	The power module detected an overheat.	PR	Dynamic stop	Coast to a stop
3C	Regeneration circuit error	An error was detected in the regenerative transistor or in the regenerative resistor.	PR	Dynamic stop	-
3D	Power supply voltage error at acceleration/deceleration	A motor control error during acceleration/deceleration, due to a power voltage failure, was detected.	PR	Dynamic stop	-
3E	Magnetic pole position detection error	The magnetic pole position, detected in the magnetic pole position detection control, is not correctly detected.	AR	Dynamic stop	Coast to a stop

(Note 1) Definitions of terms in the table are as follows.

Main side encoder: Encoder connected to CN2

Sub side encoder: Encoder connected to CN3

(Note 2) Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions. When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the NC and servo drive unit power ON again.

No.	Name	Details	Reset method	Servo stop method	Spindle stop method
41	Feedback error 3	Either a missed feedback pulse in the motor side encoder or an error in the Z-phase was detected in the full closed loop system.	PR	Dynamic stop	Coast to a stop
42	Feedback error 1	Either a missed feedback pulse in the position detection or an error in the Z-phase was detected. Or the distance-coded reference check error exceeded the allowable value when the distance-coded reference scale was used.	PR	Dynamic stop	Coast to a stop
43	Feedback error 2	An excessive difference in feedback was detected between the machine side encoder and the motor side encoder.	PR	Dynamic stop	Coast to a stop
45	Fan stop	An overheat of the power module was detected during the cooling fan stopping.	PR	Dynamic stop	Coast to a stop
46	Motor overheat / Thermal error	Either the motor or the motor side encoder detected an overheat. Or, the thermistor signal receiving circuit of the linear motor or direct-drive motor was disconnected. Or, the thermistor signal receiving circuit was short-circuited.	NR	Deceleration stop	Deceleration stop
48	Main side encoder: Error 5	An error was detected by the encoder connected to the main side. The error details are different according to the connected encoder. Refer to "Encoder alarm".		Dynamic stop	Coast to a stop
49	Main side encoder: Error 6				
4A	Main side encoder: Error 7				
4B	Main side encoder: Error 8				
4C	Current error at initial magnetic pole estimate	Current detection failed at the initial magnetic pole estimation.	NR	Dynamic stop	Coast to a stop
4D	Dual signal error	An error was detected in the signal related to the dual signal. Refer to "Dual signal error (4D)" for details.			
4E	NC command mode error	An error was detected in the control mode send from the NC.	NR	Deceleration stop	Deceleration stop
4F	Instantaneous power interrupt	The control power supply has remained shut down.	NR	Deceleration stop	Deceleration stop
50	Overload 1	Overload detection level became 100% or more. The motor or the drive unit is overloaded.	NR	Deceleration stop	Deceleration stop
51	Overload 2	In a servo system, current command of 95% or more of the unit's max. current was given continuously for 1 second or longer. In a spindle system, current command of 95% or more of the motor's max. current was given continuously for 1 second or longer.	NR	Deceleration stop	Deceleration stop
52	Excessive error 1	A position tracking error during servo ON was excessive.	NR	Deceleration stop	Deceleration stop
53	Excessive error 2	A position tracking error during servo OFF was excessive.	NR	Dynamic stop	-
54	Excessive error 3	There was no motor current feedback when the alarm "Excessive error 1" was detected.	NR	Dynamic stop	Coast to a stop
56	Commanded speed error	The encoder has detected that the commanded speed exceeded 1.15 times of the rapid traverse rate (rapid), or the motor rotation speed exceeded the maximum speed.	NR	Deceleration stop	Deceleration stop
58	Collision detection 1: G0	A disturbance torque exceeded the allowable value in rapid traverse modal (G0).	NR	Max cap dec stop	-
59	Collision detection 1: G1	A disturbance torque exceeded the allowable value in the cutting feed modal (G1).	NR	Max cap dec stop	-
5A	Collision detection 2	A current command with the maximum drive unit current value was detected.	NR	Max cap dec stop	-

(Note 1) Definitions of terms in the table are as follows.

Main side encoder: Encoder connected to CN2

Sub side encoder: Encoder connected to CN3

(Note 2) Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions. When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the NC and servo drive unit power ON again.

No.	Name	Details	Reset method	Servo stop method	Spindle stop method
5B	Safely limited: Commanded speed monitoring error	A commanded speed exceeding the safely limited speed was detected in the safely limited mode.	PR	Deceleration stop	Deceleration stop
5D	Safely limited: Door state error	The door state signal input in the NC does not coincide with the door state signal input in the drive unit in the safely limited mode. Otherwise, door open state was detected in normal mode.	PR	Deceleration stop	Deceleration stop
5E	Safely limited: Speed feedback monitoring error	A motor speed exceeding the safely limited speed was detected in the safely limited mode.	PR	Deceleration stop	Deceleration stop
5F	External contactor error	A contact of the external contactor is welding.	NR	Deceleration stop	Deceleration stop
60, 61, 63 to 77	Power supply alarm	The power supply unit detected an error. The error details are different according to the connected power supply unit. Refer to "Power supply alarm" for details.		Dynamic stop	Coast to a stop
62	Power supply: Frequency error			Deceleration stop	Deceleration stop
80	Main side encoder cable error	The cable type of the motor side encoder cable is for rectangular wave signal.	AR	Initial error	-
81	Sub side encoder cable error	The cable type of the machine side encoder cable does not coincide with the encoder type which is set by the parameter.	AR	Initial error	-
87	Drivers communication error	The communication frame between drive units was aborted.	PR	Dynamic stop	Coast to a stop
88	Watchdog	The drive unit does not operate correctly. LED display is fixed as "88". "888" is displayed for MDS-EJ/EJH Series.	AR	Dynamic stop	Coast to a stop
8A	Drivers communication data error 1	The communication data 1 between drivers exceeded the tolerable value in the communication between drive units.	PR	Dynamic stop	Coast to a stop
8B	Drivers communication data error 2	The communication data 2 between drivers exceeded the tolerable value in the communication between drive units.	PR	Dynamic stop	Coast to a stop

(Note 1) Definitions of terms in the table are as follows.

Main side encoder: Encoder connected to CN2

Sub side encoder: Encoder connected to CN3

(Note 2) Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions. When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the NC and servo drive unit power ON again.

Encoder alarm (Servo drive unit)

Alarm number when the encoder is connected to CN2 side		2B	2C	2D	2E	48	49	4A	4B
Alarm number when the encoder is connected to CN3 side		1B	1C	1D	1E	27	28	29	2A
OSA405 OSA676 OSA24RS	Mitsubishi Electric	Memory alarm	LED alarm	Data alarm	Encoder thermal error	-	-	-	-
OSA405ET2AS OSA676ET2AS		Memory alarm	LED alarm	Data alarm	Encoder thermal error	-	-	-	-
MDS-EX-HR		Memory error	-	Data error	-	Scale not connected	-	-	-
CSW26KS		Memory alarm	LED alarm	Data alarm	Encoder thermal error	-	-	-	-
AT343 AT543 AT545 AT1143 ST748	Mitsutoyo	Initialization error (bit0)	EEPROM error (bit5)	Photoelectric type, static capacity type data mismatch (bit1)	ROM/RAM error (bit6)	CPU error (bit4)	Photoelectric type overspeed (bit7)	Static capacity type error (bit3)	Photoelectric type error (bit2)
LC195M, LC495M, LC291M, LIC2197M, LIC2199M MC15 RCN2590M, RCN5390M, RCN5590M, RCN8390M ROC425M, ROC2390M ECA4000Series EIB Series	HEIDENHAIN	Initialization error (bit0)	EEPROM error (bit5)	Relative/absolute position data mismatch (bit1)	ROM/RAM error (bit6)	CPU error (bit4)	Overspeed (bit7)	Absolute position data error (bit3)	Relative position data error (bit2)
MPRZ scale	NIDEC MACHINE TOOL	Installation accuracy fault (bit4)	-	Detection position deviance (bit1)	Scale breaking (bit0)	Absolute position detection fault (bit5)	-	Gain fault (bit3)	Phase fault (bit2)
SR27A, SR67A SR85, SR74 SR84, RU77 RS87	Magnescale	Laser diode error (bit0)	System memory error (bit5)	Encoder mismatch error (bit1)	-	-	Over speed (bit7)	Absolute position data error (bit3)	Relative position data error (bit2)
SAM/SVAM/ GAM/G2AM/ LAM/G3BM/ HAM/H2AM Series	FAGOR	-	-	Absolute value detection error (bit3)	H/W error (bit1)	CPU error (bit0)	-	-	-
RL40N/RA/FORTiS Series	Renishaw	Initialization error (bit0)	-	Absolute position data error (bit3)	-	-	Over speed (bit7)	-	-
WMFA/WMBA/ WMRA/LMFA/ LMBA Series (Note)	AMO	Initialization error (bit0)	-	Relative/absolute position data mismatch (bit2)	-	-	Over speed (bit5)	Absolute position data error (bit6)	-
AMS-ABS-3B Series	Schneeberger	- (bit0)	-	-	Absolute position data error (bit3)	-	-	-	-
MHS-04B Series	GUBOA	H/W error (bit0)	Waveform error (bit1)	-	Power voltage warning (bit3)	-	-	-	-

(Note 1) A drive unit processes all reset types of alarms as "PR". However, "AR" will be applied according to the encoder.
(Note 2) Bit No. in the table refers to a bit assignation for encoder side alarm management data.

Encoder alarm (Spindle drive unit)

Alarm number when the encoder is connected toCN2 side		2B	2C	2D	2E	48	49	4A	4B
Alarm number when the encoder is connected to CN3 side		1B	1C	1D	1E	27	28	29	2A
TS5690 TS5691	Mitsubishi Electric	Memory error	Waveform error	-	-	-	Overspeed	-	Relative position data error
MDS-EX-HR		Initialization error	-	Data error	-	Connection error	-	-	-
OSA24RS		CPU error	Waveform error	Data alarm	Encoder thermal error	-	-	-	-
EIB Series	HEIDENHAIN	Initialization error (bit0)	EEPROM error (bit5)	-	-	CPU error (bit4)	Overspeed (bit7)	-	Relative position data error (bit2)
ADB-K70M	NIDEC MACHINE TOOL	Installation accuracy fault (bit4)	-	Detection position deviance (bit1)	Scale breaking (bit0)	-	-	Gain fault (bit3)	Phase fault (bit2)
GEL2449M	LENORD+BAUER	Waveform error (bit0)	Analog signal warning (bit1)	EEPROM error (bit2)	Power voltage warning (bit3)	H/W error (bit4)	Overspeed warning (bit5)	Count error (bit6)	Overheat warning (bit7)
MHS-04B Series	GUBOA	H/W error (bit0)	Waveform error (bit1)	-	Power voltage warning (bit3)	-	-	-	-

(Note 1) A drive unit processes all reset types of alarms as "PR". However, "AR" will be applied according to the encoder.
(Note 2) Bit No. in the table refers to a bit assignation for encoder side alarm management data.

Dual signal error (4D)

No.	Name	Sub info	Details
004D.xxx	Dual signal error	-	An error was detected in the signal related to the dual signal. The name of the axis with an error is displayed. The number "xxx" in the decimal place indicates the sub-number.

(Note) Resetting method may be "PR" depending on the sub-number in the decimal place.

004D.xxx : Alarm number.Sub-number

Sub-No.	Name	Details	Reset method	Axis type	Servo stop method	Spindle stop method
1	Power shutoff error	Either of the STO signals entered an input state while the STO function is disabled.	NR	Each axis	Dynamic stop	Coast to a stop
2	Illegal power shutoff error	Either of the STO signals entered an input state during servo ON command or during deceleration and stop with the STO function enabled.	NR	Each axis	Dynamic stop	Coast to a stop
3	STO signal mismatch error	Input states of two STO signals were mismatched while the STO function is enabled.	NR	Each axis	Dynamic stop	Coast to a stop
15	Safety communication: Communication error 1	A receiving error was detected in the safety communication.	NR	Each axis	Dynamic stop	Coast to a stop
16	Safety communication: Initial communication error 1	A receiving error was detected in the initial communication for the safety communication.	PR	Each axis	Dynamic stop	Coast to a stop
17	Voltage diagnosis error	A power error was detected in the safety function.	NR	Each axis	Dynamic stop	Coast to a stop
19	DRAM diagnosis error	A DRAM error was detected in the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
21	Control process error	An error was detected in the status of software execution for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
23	Safety encoder: Initial communication error 1	An error was detected in the initial communication with a safety encoder.	PR	Each axis	Initial error	Initial error
24	PCB error	A PCB error was detected in the safety function.	PR	Each axis	Initial error	Initial error
25	Synchronization error	A synchronization error was detected in the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
26	Flash ROM diagnosis error	A Flash ROM error was detected in the safety function.	PR	Each axis	Initial error	Initial error
33	Safety encoder: Communication error 1	An error was detected in the communication with a safety encoder.	PR	Each axis	Dynamic stop	Coast to a stop
34	Safety encoder: Diagnosis error 1	A power supply voltage error was detected in the safety encoder.	PR	Each axis	Dynamic stop	Coast to a stop
35	Safety encoder: Diagnosis error 2	A H/W error was detected in the safety encoder.	PR	Each axis	Dynamic stop	Coast to a stop
36	Safety encoder: Diagnosis error 3	A process error was detected in the safety encoder.	PR	Each axis	Dynamic stop	Coast to a stop
37	Safety encoder: Diagnosis error 4	An A/D conversion error was detected in the safety encoder.	PR	Each axis	Dynamic stop	Coast to a stop
39	Non-safety encoder: Position feedback fixation diagnosis error	The position feedback from the non-safety encoder remains unchanged.	PR	Each axis	Dynamic stop	Coast to a stop
40	Safety encoder: Thermal error	The safety encoder detected a thermal error.	PR	Each axis	Dynamic stop	Coast to a stop
53	Safety communication: Transmission interval mismatch error	An error was detected in the transmission interval setting.	NR	Each axis	Initial error	Initial error
54	Safety communication: Initial communication error 2	A receiving error was detected in the initial communication for the safety communication.	NR	Each axis	Initial error	Initial error
55	Safety communication: Communication error 2	A receiving error was detected in the safety communication.	NR	Each axis	Dynamic stop	Coast to a stop
56	Safety parameter setting range error	A setting error was detected in the safety parameter.	PR	Each axis	Initial error	Initial error

004D.xxx : Alarm number.Sub-number

Sub-No.	Name	Details	Reset method	Axis type	Servo stop method	Spindle stop method
57	Safety parameter combination error	A combination error was detected in the safety parameter.	PR	Each axis	Initial error	Initial error
65	Register diagnosis error	A register diagnosis error was detected in the safety function.	PR	Each axis	Initial error	Initial error
66	Calculation device diagnosis error	An error was detected in the calculation device diagnosis for the safety function.	PR	Each axis	Initial error	Initial error
67	Sequence diagnosis error	An error was detected in the sequence diagnosis for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
68	Stack diagnosis error	An error was detected in the stack diagnosis for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
69	Temperature diagnosis error	Overheat was detected in the safety function.	NR	Each axis	Dynamic stop	Coast to a stop
71	Watchdog error	The safety function is not operating correctly.	PR	Each axis	Dynamic stop	Coast to a stop
72	Clock diagnosis error	An error was detected in the clock diagnosis for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
74	DO output compare error	An error was detected in the status of DO output for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
75	ISC communication error	An error was detected in the inter-system communication for the safety function.	PR	Each axis	Dynamic stop	Coast to a stop
78	Safety communication: Initial communication error 3	A receiving error was detected in the initial communication for the safety communication.	PR	Each axis	Dynamic stop	Coast to a stop
88	Safety circuit: STO error	An error was detected in the read-back diagnosis for STO.	PR	Each axis	Dynamic stop	Coast to a stop
91	Safety circuit: SBC error	An error was detected in the read-back diagnosis for SBC.	PR	Each axis	Dynamic stop	Coast to a stop
126	Safety communication: Communication error 3	A receiving error was detected in the safety communication.	PR	Each axis	Dynamic stop	Coast to a stop

(2) Power supply alarm

No.	Name	Details	Reset method
61	Power supply: Power module overcurrent	Overcurrent protection function in the power module has started its operation.	PR
62	Power supply: Frequency error	The input power supply frequency increased above the specification range.	PR
66	Power supply: Process error	An error occurred in the process cycle.	PR
67	Power supply: Phase interruption	An open-phase condition was detected in input power supply circuit.	PR
68	Power supply: Watchdog	The system does not operate correctly. LED display is fixed as "8".	AR
69	Power supply: Grounding	The motor power cable is in contact with FG (Frame Ground).	PR
6A	Power supply: External contactor welding	A contact of the external contactor is welding.	PR
6B	Power supply: Rush circuit error	An error was detected in the rush circuit.	PR
6C	Power supply: Main circuit error	An error was detected in charging operation of the main circuit capacitor.	PR
6D	Parameter setting error	An error was detected in the parameter sent from the drive unit.	PR
6E	Power supply: H/W error	An error was detected in the internal memory.	AR
	A/D error	An error was detected in the A/D converter.	
	Unit ID error	An error was detected in the unit identification.	
6F	Power supply error	No power supply is connected to the drive unit, or a communication error was detected.	AR (Note 4)
70	Power supply: External emergency stop error	A mismatch of the external emergency stop input and NC emergency stop input continued for 30 seconds.	PR
71	Power supply: Instantaneous power interruption	The power was momentarily interrupted.	NR
72	Power supply: Fan stop	A cooling fan built in the power supply unit stopped, and overheat occurred in the power module.	PR
73	Power supply: Over regeneration	Over-regeneration detection level became over 100%. The regenerative resistor is overloaded. This alarm cannot be reset for 15 min from the occurrence to protect the regeneration resistor. Leave the drive system energized for more than 15 min, then turn the power ON to reset the alarm.	NR
74	Power supply: Option unit error	An alarm was detected in the power backup unit (power supply option unit).	NR (Note 3)
75	Power supply: Overvoltage	L+ and L- bus voltage in main circuit exceeded the allowable value. As the voltage between L+ and L- is high immediately after this alarm, another alarm may occur if this alarm is reset in a short time. Wait more than 5 min before resetting so that the voltage drops.	NR
76	Power supply: Function setting error	The rotary switch setting of external emergency stop is not correct, or a wrong external emergency stop signal is input.	AR
	Power supply: Function selection error	Undefined area for the rotary switch is selected	
77	Power supply: Power module overheat	Thermal protection function in the power module has started its operation.	PR

(Note 1) If a power supply alarm (60 to 77) occurs, all servos will stop with the dynamic brakes, and all spindles will coast to a stop.

(Note 2) "b", "C" and "d" displayed on the power supply unit's LED as a solid light (not flickering) do not indicate an alarm.

(Note 3) Check the LED display of the power backup unit to identify what alarm is occurring to the power backup unit.

**** Refer to your drive unit's instruction manual for details.**

(Note 4) When the power supply alarm (6F) is detected in the 2nd part system, the reset method differs depending on the detected power supply alarm.

3.2 Initial Parameter Errors (S02)

S02	**Initial parameter error**	2201-2264	(Axis name)

Details

The servo parameter setting data is illegal.The alarm No. is the No. of the servo parameter where the error occurred.

Remedy

Check the descriptions for the appropriate servo parameters and correct them.

Refer to "Parameter Numbers during Initial Parameter Error" of each drive unit instruction manual for details.

S02	**Initial parameter error**	2301	(Axis name)

Details

The number of constants to be used in the following functions is too large:
Electronic gears.
Position loop gain.
Speed feedback conversion.

Remedy

Check that all the related parameters are specified correctly.

sv001: PC1, sv002: PC2, sv003: PGN1

sv018: PIT, sv019: RNG1, sv020: RNG2

S02	**Initial parameter error**	2302	(Axis name)

Details

When high-speed serial incremental encoder (OSE104, OSE105) is connected, parameters for absolute position are set to ON.
Set the parameters for absolute position detection to OFF.
To detect an absolute position, replace the incremental specification encoder with an absolute encoder.

Remedy

Check that all the related parameters are specified correctly.

sv017: SPEC, sv025: MTYP

S02	**Initial parameter error**	2303	(Axis name)

Details

No servo option is found.
The closed loop (including the ball screw-end detection) or dual feedback control is an optional function.

Remedy

Check that all the related parameters are specified correctly.

sv025: MTYP/pen

sv017: SPEC/dfbx

S02	**Initial parameter error**	2304	(Axis name)

Details

No servo option is found.
The SHG control is an optional function.

Remedy

Check that all the related parameters are specified correctly.

sv057: SHGC

sv058: SHGCsp

S02	**Initial parameter error**	2305	(Axis name)

Details

No servo option is found.
The adaptive filtering is an optional function.

Remedy

Check that all the related parameters are specified correctly.

sv027: SSF1/aflt

S02	Initial parameter error:PR	13001-13256	(Axis name)

Details

Parameter error
The spindle parameter setting data is illegal.
The alarm No. is the No. of the spindle parameter where the error occurred.

Remedy

Check the descriptions for the appropriate spindle parameters and correct them.
Even when the parameter is set to a value within the setting range, an error is occurring due to the hardware compatibility or specifications or in relation to several other parameters.
Refer to "Parameter Numbers during Initial Parameter Error" of each drive unit instruction manual for details.

3.3 Safety Function Errors (S05)

S05	**Safety function error**	0001	(Axis name)

Details

The STO signal has been input through the CN8 connector.

Remedy

Make sure that a short-circuiting connector has been inserted into CN8.

S05	**Safety function error**	0002	(Axis name)

Details

STO signal is input by dedicated wiring STO function during servo ON.

Remedy

Refer to the manual of drive unit.

S05	**Safety function error**	0004	(Axis name)

Details

STO signal is illegally input by dedicated wiring STO function during servo OFF.
(Illegal input : Signal input state for STO1 and STO2 is mismatched.)

Remedy

Refer to the manual of drive unit.

S05	**Safety function error**	0006	(Axis name)

Details

STO signal is illegally input by dedicated wiring STO function during servo ON.
(Illegal input : Signal input state for STO1 and STO2 is mismatched.)

Remedy

Refer to the manual of drive unit.

3.4 Parameter Errors (S51)

S51	**Parameter error**	2201-2264	(Axis name)

Details

Servo parameter setting data is illegal.The alarm No. is the No. of the servo parameter where the warning occurred.

Remedy

Check the descriptions for the appropriate servo parameters and correct them.

S51	**Parameter error**	13001-13256	(Axis name)

Details

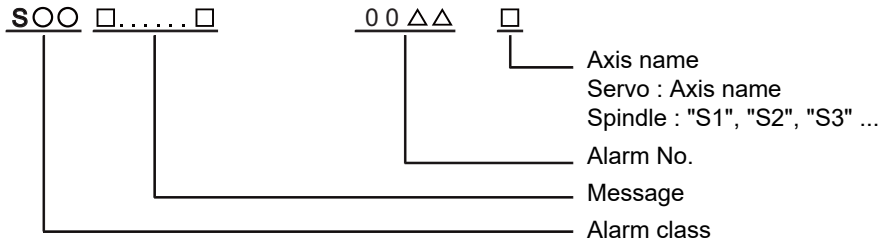
Spindle parameter setting data is illegal.
The alarm No. is the No. of the spindle parameter where the warning occurred.

Remedy

Check the descriptions for the appropriate spindle parameters and correct them.
Even when the parameter is set to a value within the setting range, an error is occurring due to the hardware compatibility or specifications or in relation to several other parameters.
Refer to "Parameter Numbers during Initial Parameter Error" of each drive unit instruction manual for details.

3.5 Servo Warnings (S52)

Servo warning is displayed in the following format.



Alarm class	Message
S52	Name or "Servo warning"

The numerical value in the column of "No." in the table after the following page is the double-digit number displayed on the LED which is on the front-side of the drive unit.

At the time of the occurrence of the servo warnings (S52), this number will be displayed in the last two digits of the "alarm number".

(Note 1) For the troubleshooting at the time of the occurrence of the servo warnings, refer to your drive unit's instruction manual.

(Note 2) When multiple servo alarms have occurred, only an alarm which occurred first is displayed on the alarm screen. Refer to the alarm history on the drive monitor screen as necessary.

(1) Drive unit warning

No.	Name	Details	Reset method	Stop method
96	Scale feedback error	An excessive difference in feedback amount was detected between the main side encoder and the MPI scale in MPI scale absolute position detection system.	*	-
97	Scale offset error	An error was detected in the offset data that is read at the NC power-ON in MPI scale absolute position detection system.	PR	-
9B	Incremental encoder/ magnetic pole shift warning	The difference between the magnetic pole position after the phase Z has been passed (magnetic pole shift amount:SV028) and the initially detected position is excessive in the built-in motor's incremental control system.The magnetic pole is controlled by the initial detection value.	PR	-
9E	Absolute position encoder: Revolution counter error	An error was detected in the revolution counter data of the absolute position encoder. The accuracy of absolute position is not guaranteed.	*	-
9F	Battery voltage drop	The battery voltage to be supplied to the absolute position encoder is dropping.	NR	-
A3	In initial setup of ABS position	This warning is detected until the axis reaches the reference position during the initial setup of the distance-coded reference check function. This warning turns OFF after the axis has reached the position, thus set the value displayed on the drive monitor to the parameter.	PR	-
A4	Dual signal warning	An input was detected in the signal related to the dual signal. Refer to "Dual signal warning (A4)" for details.	*	-
A6	Fan stop warning	A cooling fan in the drive unit stopped.	*	-
E0	Overregeneration warning	Over-regeneration detection level exceeded 80%.	*	-
E1	Overload warning	A level of 80% of the Overload 1 alarm state was detected.	*	-
E4	Parameter warning	An incorrect set value was detected among the parameters send from the NC in the normal operation.	*	-
E6	Control axis detachment warning	A control axis is being detached. (State display)	*	-
E7	NC emergency stop	In NC emergency stop. (State display)	*	Dec stop enabled
E8 to EF	Power supply warning	The power supply unit detected a warning. The error details are different according to the connected power supply unit. Refer to "Power supply warning".	*	- *EA: Dec stop enabled

(Note1) Definitions of terms in the table are as follows.

Main side encoder: Encoder connected to CN2

Sub side encoder: Encoder connected to CN3

(Note 2) Resetting methods

*** : Automatically reset once the cause of the warning is removed.**

NR: Reset with the NC RESET button. This warning can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This warning can also be reset with the AR resetting conditions. When the control axis is removed, this warning can be reset with the NC RESET button. (Excluding warning 93.)

AR: Reset by turning the NC and servo drive unit power ON again.

(Note 3) Servo and spindle motor do not stop when the warning occurs.

(Note 4) When an emergency stop is input, servo and spindle motor decelerate to a stop. (When SV048, SV055 or SV056 is set for servo and when SP055 or SP056 is set for spindle.)

Dual signal warning (A4)

No.	Name	Sub info	Details
00A4.00	Dual signal warning	Axis name	The system has been set in the STO state. The STO state is also entered at the time of emergency stop, but in this case, this warning will not appear because the emergency stop has priority.

(2) Power supply warning

No.	Name	Details	Reset method
E9	Instantaneous power interruption warning	The power was momentarily interrupted.	NR
EA	In external emergency stop state	External emergency stop signal was input.	*
EB	Power supply: Over regeneration warning	Over-regeneration detection level exceeded 80%.	*
EE	Power supply: Fan stop warning	A cooling fan built in the power supply unit stopped.	*
EF	Power supply: Option unit warning	A warning was detected in the power backup unit (power supply option unit).	* (Note 3)

(Note 1) Resetting methods

*** : Automatically reset once the cause of the warning is removed.**

NR: Reset with the NC RESET button. This warning can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This warning can also be reset with the AR resetting conditions. When the control axis is removed, this warning can be reset with the NC RESET button. (Excluding warning 93.)

AR: Reset by turning the NC and servo drive unit power ON again.

(Note 2) Servo and spindle motor do not stop when the warning occurs.

(Note 3) Check the LED display of the power backup unit to identify what warning is occurring to the power backup unit.

**** Refer to your drive unit's instruction manual for details.**

3.6 Safety Function Warnings (S53)

S53	**Safety function warning**	0001	(Axis name)

Details

The system has been set in the STO state.
The STO state is also entered at the time of emergency stop, but in this case, this warning will not appear because the emergency stop has priority.

4

MCP Alarms (Y)

Y02	**System alm: Process time over**	0050	

Details

System alarm: Process time is over.

Remedy

The software or hardware may be damaged.
Contact the service center.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	**SV commu er: Data ID error**	0051	xy03

Details

A communication error has occurred between controller and drive unit.
x: Channel No. (0 to)
y: Drive unit rotary switch No. (0 to)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	**SV commu er: Recv frame No.**	0051	xy04

Details

A communication error has occurred between controller and drive unit.
x: Channel No. (from 0)
y: Drive unit rotary switch No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	**SV commu er: Commu error**	0051	x005

Details

A communication error has occurred between controller and drive unit.
x: Channel No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	SV commu er: Connect error	0051	x006

Details

A communication error has occurred between controller and drive unit.
x: Channel No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	SV commu er: Safe posn FB err	0051	x007

Details

A communication error has occurred between controller and drive unit.
x: Channel No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	SV commu er : Init commu error	0051	xy20

Details

A communication error has occurred between controller and drive unit.
A drive unit stopped due to transition failure from initial communication to runtime.
x: Channel No. (from 0)
y: Drive unit rotary switch No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	SV commu er: Node detect err	0051	xy30

Details

A communication error has occurred between controller and drive unit.
No response from drive unit to the request from NC when setting network configuration.
x: Channel No. (from 0)
y: Station No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y02	SV commu er: Commu not support	0051	xy31

Details

A communication error has occurred between controller and drive unit.
Drive unit's software version doesn't support the communication mode that the controller requires.
x: Channel No. (from 0)
y: Station No. (from 0)

Remedy

- Take measures against noise.
- Check for any failure of the communication cable connectors between controller and drive unit or between two drive units.
- Check for any failure of the communication cables between controller and drive unit or between two drive units.
- A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and contact the Service Center.
- Update the drive unit software version.

(Note) When two or more alarms (Y02 0051) occur at the same time, only the alarm which has occurred first is displayed.

Y03	Drive unit unequipped	axis name

Details

The drive unit is not properly connected.

Axis name: NC axis drive unit not mounted

Pn: PLC axis drive unit not mounted (n = axis No.)

Sn: Spindle axis drive unit not mounted

(n = spindle No. or Sp disp name "#43107 S_disp_name")

Remedy

Check the drive unit mounting state.

- Check the end of the cable wiring.
- Check for any broken wires.
- Check the connector insertion.
- The drive unit input power has not been ON.
- The drive unit axis No. switch is illegal.

Y05	Initial parameter error	(Parameter No.)

Details

There is a problem in the value set for the number of axes or the number of part systems.

Remedy

Correct the value set for the following corresponding parameters:
"#1001 SYS_ON (System validation setup)",
"#1002 axisno (Number of axes)",
"#1039 spinno (Number of spindles)", etc.

"#2187 chgPLCax (PLC axis switchover axis No.)", etc.

- Make sure that the parameters unavailable for a switchover axis (#1068 slavno, #12800 chgauxno, etc.) are OFF.

Y05	Initial parameter error	2674	(Sub-number)

Details

The parameter setting of the multiple axis synchronization control is incorrect.

The sub-number is displayed after the error number.

0002:
More than three sets combining the synchronization control and the multiple axis synchronization control are set.

0003:
The slave axis of multiple axis synchronization control overlaps with the synchronization control axis.

0004:
The master axis of multiple axis synchronization control overlaps with the synchronization control axis.

0005:
The master axis is set to the axis which crosses the part system.

0006:
The slave axis setting is also set to the master axis.

0007:
The function which cannot be used with the multiple axis synchronization control is enabled.

Remedy

Correct the parameter setting.

Y05	Too many teach data points	12107

Details

The total number of points after summing the value set in "#12107 Teaching-DataNum" (Number of teaching data points to be allocated to each part system) for all part systems exceeds the maximum number.

Remedy

- Make sure that the total number of points does not exceed the maximum number.

Y05	Initial parameter error	90102

Details

The axis specified as the height control axis is not valid.

- The axis name does not exist.
- The axis is a rotary axis.
- The axis is the reference axis or the synchronized axis of synchronization control or multiple-axis synchronization control.
- The axis is the sub axis of simple synchronous control.
- The drive unit is not Ver.B9 or newer version of MDS-E/EH-V1 Series (servo).
- The axis is being used in NC axis/PLC axis switchover.

Remedy

Correct the value of "#90102 lsr_hight_ax" (Height control axis).

Y06	mcp_no setting error	0001

Details

There is a skipped number in the channels.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y06	mcp_no setting error	0002

Details

There is a duplicate setting for random layout.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y06	mcp_no setting error	0003

Details

The drive unit fixed setting "0000" and random layout setting "****" are both set.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y06	mcp_no setting error	0004

Details

The spindle/C axis "#1021 mcp_no (Drive unit I/F channel No. (servo))" and "#3031 smcp_no (Drive unit I/F channel No. (spindle))" are not set to the same values.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y06	mcp_no setting error	0005

Details

A random layout has been set while "#1154 pdoor" has been set to "1" in two-part system.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y06	mcp_no setting error	0006

Details

The channel No. parameter is not within the setting range.

Remedy

Check the values set for the following parameters.
"#1021 mcp_no (Drive unit I/F channel No. (servo))"
"#3031 smcp_no (Drive unit I/F channel No. (spindle))"

Y07	Too many axes connected	00xy

Details

The drive units more than the number of axes specified with the parameter are connected.
Or the number of axes connected to each channel exceeds the maximum number of connectable axes.
The exceeded number of axes per channel is displayed as alarm No.

x: Exceeded number of axes at drive unit interface channel 2 (0 to F)

y: Exceeded number of axes at drive unit interface channel 1 (0 to F)

This alarm also occurs when the drive unit is connected only with the 2nd channel without connecting with the 1st channel.

Remedy

For the channel whose alarm No. is other than "0", detach the connected axes by the number of indicated alarm No.
The number of connected axes should be less than or equal to the number specified with the parameter or the maximum number of connections.

(Note 1) The number of axes is limited per each drive unit interface channel.

(Note 2) Maximum number of axes that can be connected differs depending on whether or not an expansion unit is available or the setting of "#11012 16 axes for 1ch" (Connecting 16 axes for 1ch).
With the expansion unit, up to eight axes can be connected to a channel. Without the expansion unit, up to eight axes are allowed when "#11012 16 axes for 1ch" (Connecting 16 axes for 1ch) is set to "0", sixteen axes when set to "1".

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y08 Too many drive units connected" and "Y09 Too many axisno connected".

Y08	Too many drive units connected	00xy

Details

The number of drive units connected to each channel exceeds 8.
The exceeded number of drive units per channel is displayed as alarm No.

x: Exceeded number of drive units at drive unit interface channel 2 (0 to F)

y: Exceeded number of drive units at drive unit interface channel 1 (0 to F)

Remedy

Remove drive units from the channel whose alarm No. is other than "0" for the number displayed as the alarm No. Keep the number of connected drive units to 8 or less.

(Note 1) The drive unit is not counted when all the axes connected to it are invalid.

(Note 2) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 3) The alarm "Y07 Too many axes connected" and "Y09 Too many axisno connected" are displayed taking precedence over this alarm.

Y09	Too many axisno connected	00xy

Details

The No. of the axis (drive unit's rotary switch No.) connected to each channel is bigger than the No. allowed.
If the axis No. of each channel is bigger than the No. allowed, "1" is displayed for the alarm No.

x: "1" when the axis No. at drive unit interface channel 2 is too big

y: "1" when the axis No. at drive unit interface channel 1 is too big

Remedy

For the channel whose alarm No. is "1", keep the axis No. (drive unit's rotary switch No.) not bigger than the No. allowed.

(Note 1) The axis No. is limited per each drive unit interface channel.

(Note 2) The biggest allowed connected axis No. differs depending on whether or not an expansion unit is available or the setting of "#11012 16 axes for 1ch (Connecting 16 axes for 1ch)". The biggest connectable axis No. is as shown below.
With the expansion unit, axes No. '0' to '7' can be connected.
Without the expansion unit, axes No. '0' to '7' are allowed when '#11012 16 axes for 1ch (Connecting 16 axes for 1ch)' is set to '0', axes No. '0' to 'F' when set to '1'.

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y08 Too many drive units connected".

(Note 5) The alarm "Y07 Too many axes connected" is displayed taking precedence over this alarm.

Y12	No commu. with axis drv unit		

Details

Although the high-speed synchronous tapping option is valid, the connected drive unit doesn't support the option.

Remedy

- Replace the drive unit with that supports the option.
- Set "High-speed synchronous tapping disabled axis" parameter as disabled for the axis to which you don't use the high-speed synchronous tapping.

Y13	No commu. with sp drv unit		

Details

Although the high-speed synchronous tapping option is valid, the connected drive unit doesn't support the option.

Remedy

- Replace the drive unit with that supports the option.
- Set "High-speed synchronous tapping disabled axis" parameter as disabled for the spindle to which you don't use the high-speed synchronous tapping.

Y15	RIO connection error	0001	(Error channel)

Details

RIO station number is too large
Although a non-RIO2.0 unit is connected, 9 or a greater number is set as a station No.

Remedy

- If a non-RIO2.0 unit is connected, set the station No. to be 8 or smaller.
- If you use 9 or a greater station No., do not connect a non-RIO2.0 unit.

Y15	RIO connection error	0002	(Error channel)

Details

RIO3 connection error
A non-RIO2.0 unit is connected to RIO3.

Remedy

- Connect a RIO2.0-compatible or RIO2.0-dedicated unit.

Y15	RIO connection error	0003	(Error channel)

Details

RIO communication processing time exceeded (PC medium speed)
Due to too short DI/DO refresh cycle, the control may not refresh all the DI/DOs connected to one RIO channel at a time.

Remedy

- Correct the setting of "#1334 DI/DO refresh cycl(DI/DO refresh cycle)".
- Connect the remote IO units to RIO1, RIO2 and RIO3 dispersedly to reduce the number of RIO stations connected per RIO channel.

Y15	RIO connection error	0004	(Error channel)

Details

RIO communication processing time exceeded (PC high speed)
Due to too short DI/DO refresh cycle, the control may not refresh all the DI/DOs connected to one RIO channel at a time.

Remedy

- Connect the remote IO units to RIO1, RIO2 and RIO3 dispersedly to reduce the number of RIO stations (which perform high-speed input/output) per RIO channel.

Y15	**Laser I/F unit connect error**	0005	

Details

An error was detected for the connection of the laser I/F unit.

- The laser I/F unit and another RIO device have been connected to the same channel.
- The laser I/F unit has been connected to the station connected to the operation panel.
- More than one laser I/F unit is connected.
- Laser I/F unit has been failed.

Remedy

- Check the connection of the laser I/F unit.
- Replace the laser I/F unit.

Y15	**Too many RIO channels in use**	0006	

Details

Five or more RIO connectors of the NC are in use.

Remedy

- Modify the RIO configuration so that up to four RIO connectors are in use.

Y16	**Variable torsion comp invalid**		

Details

Although the variable full-closed torsion compensation function is enabled, the connected drive unit does not support the function.

Remedy

- Upgrade the drive unit version to be compatible with variable full-closed torsion compensation.
- When not using the variable full-closed torsion compensation, set "#14301 valVDC" to "0".

Y20	**Parameter compare error**	0001	(Axis name)

Details

The speed monitoring parameter in the NC does not correspond to the parameter transmitted to the drive unit.
The name of the axis with an error is displayed.

Remedy

The NC or the servo drive unit may be damaged.
Contact the service center.

Y20	**Sfty obsrvation: Cmd spd err**	0002	(Axis name)

Details

The speed exceeding the speed set with the parameter was commanded during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

Check the speed monitoring parameter and the sequence program.
Restart the NC.

Y20	**Sfty obsrvation: FB pos err**	0003	(Axis name)

Details

The commanded position, transmitted to the servo drive unit from NC, is totally different from the feedback position received from the servo drive unit during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

The NC or the servo drive unit may be damaged.
Contact the service center.

Y20	**Sfty obsrvation: FB speed err**	0004	(Axis name)

Details

Actual rotation speed of the motor is exceeding the speed that has been set with speed monitoring parameter during the speed monitoring mode.
The name of the axis with an error is displayed.

Remedy

Correct the speed observation parameter and the sequence program.
Restart the NC.

Y20	**Door signal: Input mismatch**	0005	Door No.

Details

Door state signals on the NC side and the drive side do not match. It may be caused by the followings:

- Cable disconnection
- Damaged door switch
- Damaged NC or servo drive unit

Remedy

Check the cable.
Check the door switch.
Restart the NC.

Y20	**No speed observation mode in door open**	0006	Door No.

Details

The door open state was detected when the speed monitoring mode was invalid.
The causes may be same as the ones for 0005 (Door signal: Input mismatch). Also the sequence program may not be correct.

Remedy

Correct the sequence program.
Restart the NC.

Y20	**Speed obsv: Para incompatible**	0007	(Axis name)

Details

Two speed monitoring parameters are not matched at the rising edge of the "speed monitor mode" signal.
The name of the axis with an error is displayed.

Remedy

Correct the relevant parameters so that the two speed monitoring parameters match.
Restart the NC.

Y20	**Contactor welding detected**	0008	Contactor info

Details

Contactor welding was detected.
Displays the bit corresponding to the No. of the abnormal contactor for the contactor information.
Some contactors take a while to be shutdown after the servo ready is turned OFF, and the servo ready was turned ON in the meantime.

Remedy

- Make sure that contactor's auxiliary b contact signal is output correctly to the device set on "#1330 MC_dp1(Contactor weld detection device 1)" and "#1331 MC_dp2(Contactor weld detection device 2)".
- If welding, replace the contactor.
- Restart the NC.

Y20	**No spec: Safety observation**	0009

Details

"#2313 SV113 SSF8/bitF (ssc SLS (Safely Limited Speed) function)" and "#13229 SP229 SFNC9/bitF (ssc SLS (Safely Limited Speed) function)" are set for a system with no safety observation option.

Remedy

Disable "#2313 SV113 SSF8/bitF (ssc SLS (Safely Limited Speed) function)" and "#13229 SP229 SFNC9/bitF (ssc SLS (Safely Limited Speed) function)".
Then, restart the NC.

Y20	**SDIO connector input volt err**	0010

Details

24VDC power is not supplied to SDIO connector correctly. (SDIO 4A pin supply voltage was dropped to 16V or less, or 1ms or more instant power interrupt was detected.)
In this case, "Pw sply:Inst pw interpt(DC24V)" alarm occurs because the contactor control output signal cannot be controlled.
This state remains until restarting the NC even if the cause of the alarm has been removed.

Remedy

Check the wiring. Supply 24VDC power to the SDIO connector.
Restart the NC.

Y20	**Device setting illegal**	0011	Contactor info

Details

- The device set in the parameter "#1353 MC_ct1" (Contactor shutoff output 1 device) does not exist.
- The device set in the parameter "#1353 MC_ct1" (Contactor shutoff output 1 device) is used as an output device in PLC program.

Remedy

- In the parameter "#1353 MC_ct1" (Contactor shutoff output 1 device), set the device to which a remote I/O is connected. Use the device to control the contactor.
- Confirm that the devices set by the parameter "#1353 MC_ct1" (Contactor shutoff output 1 device) are not used as an output device in PLC program.
- Restart the NC.

Y20	**Contactor operation abnormal**	0012	Contactor info

Details

Contactor's operation is not following the NC's commands.

Displays the No. of the abnormal contactor for the contactor information.

Remedy

- Check and correct "#1353 MC_ct1" (Contactor shutoff output 1 device) setting.
- Check the wiring for contactor shutoff.
- Check for contactor's welding.
- Restart the NC.

Y20	**STO function operation illegal**	0013	

Details

The drive unit's STO function has failed to work properly.

Remedy

If this alarm has occurred alone, a drive unit failure can be suspected.
If other alarms have been generated at the same time, it is also possible that there is communication problem. Check the optical cable wiring.

Y20	**STO function illegal at pwr ON**	0014	

Details

The motor power has not been shut down with the STO function when the NC power was turned ON.

Remedy

If this alarm has occurred alone, a drive unit failure can be suspected.
If other alarms have been generated at the same time, it is also possible that there is communication problem. Check the optical cable wiring.

Y20	**Dual signal: parameter setting error**	0027	

Details

A setting of #2118 SscDrSel, #3071 SscDrSelSp, #2180 S_DIN, or #3140 S_DINSp is not correct.

Remedy

- Correct the parameter setting.

Y20	**Safety observation: parameter memory error**	0031	(Parameter No.)

Details

The following parameters are not consistent with the check data.
#2180 S_DIN, #3140 S_DINSp

Remedy

- Correct the parameter setting.
- Restore the backup data, as the parameter or check data may be corrupted.

Y21	**Speed obsv signal: Speed over**	0001	(Axis name)

Details

The speed exceeds the safety speed limit when the "speed monitor mode" signal is ON.
The name of the axis with an error is displayed.

Remedy

Decelerate the speed to reset the warning and start the speed monitor.

Y40	Machine group-based stop	

Details

A machine group-based alarm stop has occurred, or the machine group-based PLC interlock signal has been input.

Remedy

♦Remove the cause of the stop by alarm.Turn OFF the machine group-based PLC interlock signal.

Y51	Parameter G0tL illegal	0001

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2004 G0tL (G0 time constant (linear))".

Y51	Parameter G1tL illegal	0002

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2007 G1tL (G1 time constant (linear))".

Y51	Parameter G0t1 illegal	0003

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2005 G0t1 (G0 time constant (primary delay) / Second-step time constant for soft acceleration/deceleration)".

Y51	Parameter G1t1 illegal	0004

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct "#2008 G1t1 (G1 time constant (primary delay)/Second-step time constant for soft acceleration/deceleration)".

Y51	Parameter grid space illegal	0009

Details

The grid space is illegal.

Remedy

Correct "#2029 grspc(Grid interval)".

Y51	Parameter stapt1-4 illegal	0012

Details

The time constant has not been set or exceeded the setting range.

Remedy

Correct the parameters from "#3017 stapt1(Tap time constant (Gear: 00))" to "#3020 stapt4(Tap time constant (Gear: 11))".

Y51	Slave axis No. illegal	0014

Details

In the axis synchronization, parameter settings for slave axis have been attempted in different part system from that of master axis.

Remedy

Correct the "#1068 slavno (Slave axis number)" setting.

Y51	**Parameter skip_tL illegal**	0015

Details

The time constant has exceeded the setting range.

Remedy

Correct "#2102 skip_tL (Skip time constant linear)".

Y51	**Parameter skip_t1 illegal**	0016

Details

The time constant has exceeded the setting range.

Remedy

Correct "#2103 skip_t1 (Skip time constant primary delay / Second-step time constant for soft acceleration/deceleration)".

Y51	**Parameter G0bdcc illegal**	0017

Details

"#1205 G0bdcc (Acceleration and deceleration before G0 interpolation)" for the 2nd part system is set to acceleration/deceleration before G0 interpolation.

Remedy

Correct "#1205 G0bdcc (Acceleration and deceleration before G0 interpolation)".

Y51	**OMR-II parameter error**	0018

Details

An illegal setting was found in the OMR-II-related parameters. OMR-II has been disabled.

Remedy

Correct the related parameter settings.

Y51	**PLC indexing stroke length err**	0019

Details

"#12804 aux_tleng (Linear axis stroke length)" has not been set or exceeded the setting range while the linear axis equal indexing is enabled for the PLC indexing axis.

Remedy

Correct "#12804 aux_tleng (Linear axis stroke length)".

Y51	**Hi-acc time const unextendable**	0020

Details

High-accuracy acceleration/deceleration time constant extension option is unavailable.

Remedy

- Adjust the setting of "#1207 G1btL" to be within the range of when the high-accuracy control time constant extension option is OFF.
- High-accuracy acceleration/deceleration time constant extension option is unavailable for a system configured with multiple part systems. Change the system to be made up of a single part system, or set the said option to OFF.

Y51	**Superimpos linear G0 error**	0022

Details

The time constant has not been set or exceeded the setting range.

Remedy

Check "#2092 plG0tL G0 time constant for superimposition control (linear)".

Y51	**Superimpos linear G1 error**	0023

Details

The time constant has not been set or exceeded the setting range.

Remedy

Check "#2094 plG1tL G1 time constant for superimposition control (linear)".

Y51	**Primary delay G0time const err**	0028

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2093 plG0t1 G0 time constant for superimposition (primary delay)/2nd step of soft acceleration/deceleration".

Y51	**Primary delay G1time const err**	0029

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2095 plG1t1 G1 time constant for superimposition (primary delay)/2nd step of soft acceleration/deceleration".

Y51	**Jerk filter time constant err**	0030

Details

Setting of "#12051 Jerk_filtG1" is greater than that of "#1568 SfiltG1".
Or setting of "#12052 Jerk_filtG0" is greater than that of "#1569 SfiltG0".

Remedy

Change the setting of "#12051 Jerk_filtG1" to be smaller than "#1568 SfiltG1".
Or change the setting of "#12052 Jerk_filtG0" to be smaller than "#1569 SfiltG0".

Y51	**Unable to alloc. hi-acc buffer**	0031

Details

The high-accuracy acceleration/deceleration buffer has failed to be allocated.

Remedy

The software or hardware may be damaged. Contact the service center.

Y51	**Too many hi-speed/accu systems**	0032

Details

The parameter "#8040 High-SpeedAcc" is set to 1 for four or more part systems.

Remedy

Set the parameter "#8040 High-SpeedAcc" to 1 for up to three part systems.

Y51	**Parameter G0tL_2 illegal**	0033

Details

The time constant is out of the specified range.

Remedy

Correct "#2598 G0tL_2 (G0 time constant 2 (linear))".

Y51	**Parameter G0t1_2 illegal**	0034

Details

The time constant is out of the specified range.

Remedy

Correct "#2599 G0t1_2 (G0 time constant 2 (primary delay)/Second-step time constant for soft acceleration/deceleration)".

Y51	**3ax line accel G0time const er**	0035

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2622 pl3G0tL G0 time constant (linear) for 3-axis serial superimposition control".

Y51	**3ax line accel G1time const er**	0036

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2624 pl3G1tL G1 time constant (linear) for 3-axis serial superimposition control".

Y51	**3ax prim delay G0time const er**	0037

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2623 pl3G0t1 G0 time constant (primary delay) for 3-axis serial superimposition control/2nd step of soft acceleration/deceleration".

Y51	**3ax prim delay G1time const er**	0038

Details

The time constant has not been set or the set time constant is out of the specified range.

Remedy

Correct "#2625 pl3G1t1 G1 time constant (primary delay) for 3-axis serial superimposition control/2nd step of soft acceleration/deceleration".

Y51	**Machine group No. discrepancy**	0039

Details

The machine group Nos. that are used for the machine groupwise alarm stop function are different among the axes related to inclined axis control and synchronous control.

Remedy

Give an identical machine group No. to all the axes related to inclined axis control and synchronous control.

Y51	**M-group alarm stop disabled**	0040

Details

The machine group-based alarm stop function has been disabled, because both the machine group-based alarm stop and collision detection functions were enabled.

Remedy

- Disable the collision detection function if you wish to use the machine group-based alarm stop function.

Y51	**Basic axes I, J, K error**	0045

Details

The 1st letter of name extension axis for 2-letter axis (#1013 axname) is specified to the basic axes I, J, and K (parameter #1026 to #1028).

Remedy

- Set the letter other than the 1st letter of name extension axis for 2-letter axis (#1013 axname) to the basic axes I, J and K (parameter #1026 to #1028).

Y51	**Values of PC1/PC2 too large**	0101

Details

The PC1 and PC2 settings for the rotary axis are too large.

Remedy

Correct "#2201 SV001 PC1 (Motor side gear ratio)" and "#2202 SV002 PC2 (Machine side gear ratio)".

Y51	**ABS/INC same axis name**	0104

Details

In the absolute/incremental command method using the axis address (L system and when "#1076 AbsInc" = 1), the axis that the same axis address is set in "#1013 axname" and "#1014 incax" exists.

Remedy

Correct the settings so that the same axis address is not set in "#1013 axname" and "#1014 incax".

Y51	Axis name initial setting err	0105

Details

The axis address that is out of the setting range (other than X, Y, Z, U, V, W, A, B, C, H) is set in "#1013 axname" or "#1014 incax".

Remedy

Set the axis address within the setting range to "#1013 axname" and "#1014 incax".

Y51	Invalid parameter combination	0108

Details

Combination of parameter settings is invalid.

Functions that cannot be used together are set to be enabled at the same time.

Remedy

Correct the parameter settings.

Select the parameter combination from the options below, and then disable one of the parameters:

- "#1268 ext04/bit6" (Tool length compensation behavior (M system only))
  "#1255 set27/bit5" (Tool length offset for multiple axis synchronization control)
- "#1268 ext04/bit6" (Tool length compensation behavior (M system only))
  "#1756 cfgPR06/bit5" (Tool length offset by tool number allocated with #12064)
- "#1274 ext10/bit3" (Tool offset number H stand-alone command (M system only))
  "#1247 set19/bit0" (Movement by tool length compensation command (M system only))
- "#1274 ext10/bit3" (Tool offset number H stand-alone command (M system only))
  "#1255 set27/bit5" (Tool length offset for multiple axis synchronization control)
- "#1274 ext10/bit3" (Tool offset number H stand-alone command (M system only))
  "#1756 cfgPR06/bit5" (Tool length offset by tool number allocated with #12064)
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount (M system only))
  "#1247 set19/bit0" (Movement by tool length compensation command (M system only))
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount (M system only))
  "#1255 set27/bit5" (Tool length offset for multiple axis synchronization control)
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount (M system only))
  "#1756 cfgPR06/bit5" (Tool length offset by tool number allocated with #12064)
- "#1227 aux11/bit1" (Select H or D code)
  "#1756 cfgPR06/bit5" (Tool length offset by tool number allocated with #12064)

Y60	Encoder communication error	0001

Details

An error was detected in the spindle encoder during spindle control with pulse train output.

Remedy

- Check for spindle encoder problems such as disconnection.

Y90	No spindle signal	0001-0007

Details

There is an error in the spindle encoder signal.
The data transmission to the drive unit is stopped when this error occurs.

Remedy

Check the spindle encoder's feedback cable and the encoder.

5

System Alarms (Z)

Z02	System error	

Details

The operation result is illegal.

Remedy

♦Contact the service center.

Z11	CC-L IE F communication error	n1 n2

Details

A communication error has occurred in the communication that uses a CC-Link IE field network unit.

n1: Indicates the No. of slot to which the expansion unit with the communication error or diagnostic alarm is mounted (in hexadecimal format).

n2: Indicates the alarm No. of the slot where the communication error or diagnostic alarm is occurring (in hexadecimal format).

Remedy

For details of the alarm number, refer to the list of messages in "CC-Link IE field network specification manual".

Z13	CC-L IE F parameter error	(Parameter No.)

Details

There is an incorrect parameter setting.

Parameter setting of #[Parameter No.] is incorrect.

Remedy

Correct the setting.

Refer to the list of messages in "CC-Link IE field network specification manual".

Z14	CC-Link IE F H/W test status	n1 n2

Details

H/W test is completed for the CC-Link IE field network unit.

Check the test result.

n1: Indicates the H/W test result of slot 1 (in hexadecimal format).

n2: Indicates the H/W test result of slot 2 (in hexadecimal format).

0: H/W test is normally completed or not performed

2: External self-loopback test is completed with connection error

3: External self-loopback test is completed with communication error

FFFF: H/W test is underway

Remedy

Take the following measures according to the test result.

0: Check the communication mode.
If the mode is H/W test, switch it to online, and turn OFF and ON the NC power.
If a mode other than H/W test is active, check the result of the other slot and take the measures.

2: Make sure the connection of Ethernet cable, or exchange the cable. And then perform the test again.
If the error occurs again, exchange the H/W.

3: Exchange the Ethernet cable, and perform the test again.
If the error occurs again, exchange the H/W.

FFFF: Turn OFF and ON the NC power.
If the result is unchanged, exchange the H/W.

Z15	CC-L IE F Basic(M) comm. error	n1 n2

Details

A communication error has occurred in the master station of CC-Link IE Field Network communication.

n1: indicates the No. of station where a communication error is occurring (in decimal notation).

0: Local station (master station)

Other than 0: Station No. of communication error

n2: indicates the alarm No. of the station where a communication error is occurring (in hexadecimal notation).

Remedy

For details of the alarm number, refer to the list of messages in "CC-Link IE Field Network Basic specifications manual".

Z16	CC-L IE F Basic(R) comm. error	n1

Details

A communication error has occurred in the remote station of CC-Link IE Field Network communication.

n1: indicates the alarm No. (in hexadecimal notation).

Remedy

For details of the alarm number, refer to the list of messages in "CC-Link IE Field Network Basic specifications manual".

Z17	CC-L IE F Basic parameter err.	#[Parameter No.]

Details

There is an incorrect parameter setting.

Parameter setting of #[Parameter No.] is incorrect.

Remedy

Correct the setting.

Refer to the list of messages in "CC-Link IE Field Network Basic specifications manual".

Z18	Network expansion card error	n1 n2

Details

An abnormality has occurred on the field network communication expansion unit.

n1: Error slot No.

n2: Error No.

Remedy

0 99:

Two Fieldbus communication expansion units which cannot be used simultaneously are mounted simultaneously. Remove either of Fieldbus communication expansion unit.

Other than 0 99:

Hardware may be faulty. Replace the field network communication expansion unit.

Z21	PROFIBUS parameter error	(Parameter No.)

Details

The parameter #[Parameter #] is incorrect.

Remedy

Correct the setting.

Z23	AXIS EX-ADR. ERROR

Details

The axis not extending the address (1 character axis) exists at the back of the axis extending the address (2 character axis) in the same system.

Remedy

- Improve the axis composition. The axis not extending the address can not exist at the back of the axis extending the address in the same system.

Z25	Unable to start safe spd clamp	

Details

The SLS observation request signal has been turned OFF in any of the following machining modes: Thread cutting, synchronous tapping, cross machining, superimposition control, tool center point control or SSS control

Remedy

- Perform the operation with the SLS observation request signal ON.
- Perform the operation with the door closed.

Z26	NC unit replacement illegal	

Details

NC unit is replaced to the FCU8-MU541, FCU8-MA541, FCU8-MU501, or FCU8-MU502.

Remedy

- Contact to your service center.
- Press the reset to cancel the warning and continue the operations.

Z27	FnctnlSafetyExpnsCard MountErr	

Details

A functional safety expansion card is mounted to the NC unit (FCU8-MU501 or FCU8-MU50).

Remedy

- Contact to your service center.
- Press the reset to cancel the warning and continue the operations.

Z28	EtherNet/IP parameter error	(Parameter No.)

Details

Parameter setting of #[Parameter No.] is incorrect.

Refer to the list of messages in" EtherNet/IP specification manual".

Remedy

Correct the setting.

Z29	EtherNet/IP communication err	n1 n2 n3 n4

Details

A communication error has occurred in the communication that uses an EtherNet/IP expansion unit.

[n1: Output the device Number (in hexadecimal format)]

FF: Simultaneously mounting two EtherNet/IP expansion units error

Other than FF: Device No. of the connected device

[n2: Output the position No. (in hexadecimal format)]

FF: Simultaneously mounting two EtherNet/IP expansion units error

Other than FF: Device No. of the connected device

[n3: Output the device communication status (IN) (in hexadecimal format)]

[n4: Output the device communication status (OUT) (in hexadecimal format)]

Remedy

For details, refer to "EtherNet/IP Specifications manual".

Z31	Socket open error(socket)	0001

Details

Socket open error (socket)

Remedy

Set the parameter then turn the power OFF and ON again.

Z31	Socket bind error(bind)	0002

Details

Socket bind error (bind)

Remedy

Set the parameter then turn the power OFF and ON again.

Z31	**Connection wait queue error(listen)**	0003

Details

Connection wait queue error (listen)

Remedy

Set the parameter then turn the power OFF and ON again.

Z31	**Connection request error(accept)**	0004

Details

Connection request error (accept)

Z31	**Data recv error(socket error)**	0005

Details

Data receive error (socket error)

Z31	**Data recv error(data error)**	0006

Details

Data receive error (data error)

Z31	**Data send error(socket error)**	0007

Details

Data send error (socket error)

Z31	**Data send error(data error)**	0008

Details

Data send error (data error)

Z31	**Socket close error(close)**	000A

Details

Socket close error (close)

Remedy

Set the parameter then turn the power OFF and ON again.

Z34	**DeviceNet error**

Details

Any of the following errors has occurred in the DeviceNet unit.

- Master function error (X03 is ON)
- Slave function error (X08 is ON)
- Message communication error (X05 is ON)

If the errors have occurred in more than one unit, the error No. of the unit with the smallest slot No. is displayed.

If the master function, slave function and message communication errors have occurred at the same time, the error is displayed in the following priority order.

1. Master function error
2. Slave function error
3. Message communication error

Remedy

- Select the [Ext. PLC link control] menu on the maintenance screen to open the unit confirmation screen, and check the unit in error and details to cancel the error.

Z35	**Direct Socket connection error**	0001

Details

- Connection has failed.
- Five or more clients attempted a connection.

Remedy

- Check the connection of the network cables, and check for broken wires and a failure of the network connection devices such as hub.
- When using the Direct Socket communication I/F, connect up to four clients.

Z35	Direct Socket receive error	0002

Details

Receiving data from a client has failed.

Remedy

- Check the connection of the network cables, and check for broken wires and a failure of the network connection devices such as hub.

Z35	Direct Socket send error	0003

Details

Sending data to a client has failed.

Remedy

- Check the connection of the network cables, and check for broken wires and a failure of the network connection devices such as hub.

Z35	Direct Socket timeout error	0004

Details

There was no response from client computers, and a timeout error occurred.

Remedy

- Check the connection of the network cables, and check for broken wires and a failure of the network connection devices such as hub.

Z35	Direct Socket comm OFF	0005

Details

The direct Socket communication I/F is OFF.

Remedy

- Check the parameter "#11051 Direct Socket ON".

Z36	EcoMonitorLight comm. error

Details

An error has occurred in the communication with EcoMonitorLight.

Remedy

Make sure that the CNC has the same communication settings (station No.,baud rate, parity and stop bit) as the EcoMonitorLight in error.

Make sure that there are no problems with the serial cable connected to the EcoMonitorLight.

Make sure to place the serial cable in a low-noise environment.

Z37	EcoMonitorLight qty discrepant

Details

The number of EcoMonitorLight units connected is inconsistent with the setting of the parameter #11061.

Remedy

Make sure that the value set in the parameter #11061 coincides with the number of EcoMonitorLight units connected to the CNC.
Also make sure all the EcoMonitorLight units connected are powered ON and the station No. is not duplicated.

Z38 Insulation deterioration wrn

Details

The motor insulation has been deteriorated.

0001: Motor insulation deterioration: Caution

0002: Motor insulation deterioration: Replacement required

0003: Motor insulation deterioration: Relay welding detected

Remedy

0001: The insulation resistance value of the motor has dropped.
Contact the service center.

0002: The insulation resistance value of the motor is less than or equal to the reference value.
Contact the service center.
When measuring the insulation resistance of the motor and the measured resistance value is less than or equal to the specified value, the motor may be damaged.

0003: The drive unit may be damaged.
Contact the service center.

Z39 E-mail send error

Details

E-mail sending with the email notification to operator function failed.

Remedy

- Check the e-mail environment setting.
- Check the registered e-mail address.
- Set "0" in "#8134 E-mail send disabled".

Z40 Format mismatch

Details

Although SRAM open parameters were changed, formatting has not been performed yet.

Remedy

- Perform formatting and restart the NC.

Z41 Decryption code is missing

Details

Decryption code for the system lock function is not yet entered, though the encryption key has been entered.

Remedy

- If you want to enable system lock, enter the decryption code.
  If you do not want, simply turn the power OFF and ON.

Z45 HD mode disabled

Details

You have set "#11053 UserProgramStorage" (User program storage destination) to "1" although the HD operation option is OFF.

Remedy

- If you store user programs in the memory, set "#11053 UserProgramStorage" to "0" and then restart the NC.
- If you store user programs in the hard disk, add the HD operation option.

Z49 RIO watchdog error 0001

Details

An error has occurred in the remote I/O unit.

Remedy

- Turn the power ON again.
- If this error remains active after the power ON, replace the remote I/O unit.

Z51	E2PROM error	001x

Details

[Type]

Z51 E2PROM error 0011: Read error

Z51 E2PROM error 0012: Write error

Remedy

- If the same alarm is output by the same operation, the cause is an H/W fault. Contact the Service Center.

Z52	Low battery	00xy

Details

Low battery voltage has been detected in the control unit or the personal computer unit.
(The battery for retaining internal data)
x: Personal computer unit
y: Control unit
1: Battery warning
3: Battery alarm

(Note) The message "Battery warning" disappears on CNC reset, but the warning remains active until the battery is replaced.

Remedy

- Replace the battery of the control unit or the personal computer unit.
- Check for disconnection of the battery cable.
- After treating the battery, check the machining program.

Z53	CNC overheat

Details

The controller or operation board temperature has risen above the designated value.

(Note) Temperature warning
When an overheat alarm is detected, the alarm is displayed and the overheat signal is output simultaneously. Automatic operation will be continued, while restarting after resetting or stopping with M02/M30 is not possible. (Restarting after block stop or feed hold is possible.)
The alarm will be cleared and the overheat signal will turn OFF when the temperature drops below the specified temperature.
Z53 CNC overheat 000x
[000x]
(For all models)
0001: The temperature in the control unit is high.
The ambient temperature must be lowered immediately when a "Z53 CNC overheat" alarm occurs. However, if the machining needs to be continued, set "#6449/bit7 Control unit temperature alarm ON" to "0". Then the alarm will be invalidated.

Remedy

- Cooling measures are required.
- Turn OFF the controller power, or lower the temperature with a cooler, etc.

Z55	RIO communication stop

Details

An error occurs in the communication between the control unit and remote I/O unit.

- Disconnection of a cable
- Fault in remote I/O unit
- Fault of power supply to remote I/O unit

The communication interrupted station is displayed in hexadecimal for each RIO channel when an error occurs in the communication between the control unit and remote I/O unit.

When the control displays a communication stop station, it divides the RIO channel stations (up to 64 stations) into groups of 8 stations. If any of the 8 stations has the communication stop error, the number is displayed with the alarm.

Z55 RIO communication stop __ __ __ __ __ __ __ __
(a) (b) (c) (d) (e) (f) (g) (h)

(a)(b): RIO1

(c)(d): RIO2

(e)(f): RIO3

(g)(h): RIO4

(a)(b), (c)(d), (e)(f), and (g)(h) indicate the following stations in hexadecimal.

bit0: 1st to 8th stations

bit1: 9th to 16th stations

bit2: 17th to 24th stations

bit3: 25th to 32nd stations

bit4: 33rd to 40th stations

bit5: 41st to 48th stations

bit6: 49th to 56th stations

bit7: 57th to 64th stations

(Example) Stations #1 and #8 of RIO1 and stations #24 and #64 of RIO3 have the communication error.
Z55 RIO communication stop 0100 8400

The RIO error status can be monitored through the R registers on the self diagnostic or I/F diagnostic screen.

Remedy

- Check and replace the cables.
- Replace the remote I/O unit.
- Check the power supply (existence of supply and voltage).

Z57	System warning

Details

Program memory capacity has been set over the value that can be formatted.
An expansion device/expansion cassette has not mounted after formatting.
The mounted expansion device/expansion cassette is different from the one that was mounted at formatting.

Remedy

Check the followings.

- Program memory capacity
- Mounting of an expansion device/expansion cassette
- APLC release option

Z57	System warning	1000

Details

- Invalid function table name for motion control release.
- Required function is missing from the function table for motion control release.
- The add-on module for motion control release cannot be decoded.

Remedy

Check the states of the following items.

- Motion control release definition file
- Add-on module

Z58 ROM write not completed

Details

The machine tool builder macro program has not been written to ROM after being edited or input.

Remedy

•Write the machine tool builder macro program to ROM.

(*) If the changes made by editing, etc. can be discarded when the NC is powered off, writing to ROM is not necessary.

Z59 Acc/dec time cnst too large

Details

Acceleration and deceleration time constants are too large.
(This alarm occurs with the stop code (T02 0206).)

Remedy

•Set the larger value for "#1206 G1bF(Maximum speed)".
•Set the smaller value for "#1207 G1btL(Time constant)".
•Set the lower feedrate.

Z60 Fieldbus communication error n1 n2 n3 n4

Details

A communication error has occurred on the Fieldbus communication with FCU8-WN563.

[n1: denotes the master station status (in the hexadecimal form).]
00: Offline: In initialization
40: Stop: I/O communication stopped
80: Clear: Resetting the output data of each slave by sending zero data
C0: In operation: In I/O communication
FF: Two Fieldbus expansion units are mounted.

[n2: denotes the error condition (in the hexadecimal form).]
bit0: Control error: Parameter error
bit1: Auto clear error: Communication with all the slave stations was cut because a communication with one slave station had an error.
bit2: Non-exchange error: A slave station has communication error.
bit3: Fatal error: Communication cannot be continued because of severe network failure.
bit4: Not ready: CNC communication is not ready.
bit5: Timeout error: Timeout is detected in communication with each station.
bit6: Not used
bit7: Not used
FF: Two Fieldbus expansion units are mounted.

[n3: denotes the slave station where communication error has occurred(in the hexadecimal form).]
(*) FF is indicated when two Fieldbus expansion units are mounted.

[n4: denotes the error number (in the hexadecimal form).]
This shows the communication state with slave station where error is occurring.

Remedy

For details, refer to "PROFIBUS-DP Specification manual".

Z64 Valid term soon to be expired xx

Details

The valid term will be expired in less than a week. Remaining valid term is xx days.

Remedy

•Obtain a decryption code from the machine tool builder and input it in the NC, then turn the power ON again.

Z65 Valid term has been expired

Details

The valid term has been expired with no decryption code input.

Remedy

•Obtain a decryption code from the machine tool builder and input it in the NC, then turn the power ON again.

Z67	**CC-Link communication error**	

Details

A communication error occurred during CC-Link communication using CC-Link unit.

Remedy

♦Refer to the list of messages in "CC-Link (Master/Local) Specification manual".

Z68	**CC-Link unconnected**	

Details

A cable between CC-Link unit and a device is disconnected or broken.

Remedy

♦Find the unconnected cable by checking SW0080 to SW0083 and connect it.

Z69	**External link error**	2

Details

A FROM/TO instruction was used while the MELSEC-Q interface expansion module is not installed.

Remedy

Install the MELSEC-Q interface expansion module.

Z69	**External link error**	3

Details

A negative value was set for an I/O No. in the FROM/TO instruction.

Remedy

Correct the I/O No.

Z69	**External link error**	4

Details

A negative value was set for transfer size in the FROM/TO instruction.

Remedy

Correct the transfer size.

Z69	**External link error**	5

Details

The number of FROM/TO instructions within one scan has exceeded 50.

Remedy

Correct the user PLC (ladder sequence) so that the number of FROM/TO instructions per scan is 50 or less.

Z69	**External link error**	6

Details

The access to the buffer memory by the FROM/TO instruction has exceeded 12K words per scan.

Remedy

Correct the user PLC (ladder sequence) so that the buffer memory access by the FROM/TO instruction won't exceed 12K words per scan. (The total size of FROM/TO is up to 12K words.)

Z69	**External link error**	7

Details

A FROM/TO instruction was used in high-speed processing.

Remedy

Delete the FROM/TO instruction from high-speed processing.

Z69	**External link error**	8

Details

The bit device number designated in the FROM/TO instruction is not a multiple of 16.

Remedy

Correct the bit device number designated in the FROM/TO instruction to be a multiple of 16.

Z69	External link error	9

Details

With a FROM/TO instruction, a value out of the address range (negative value, or 0x8000 or over) was set as the head address of the buffer memory.

Remedy

Correct the head address of the buffer memory.

Z69	External link error	10

Details

An alarm occurred in the MELSEC module mounted on the extension base.

Remedy

Check for any disconnection of the MELSEC module and the cables on the extension base. Then turn the CNC's power ON again.

Z69	External link error	11

Details

The I/O No. designated in the FROM/TO instruction is different from the mounted location of the intelligent function module on the extension base (the module's I/O No.).

Remedy

Correct the I/O No. Then turn the CNC's power ON again.

Z82	3D machine interference/No machine model	0001

Details

Machine model is not registered.

Remedy

- Press RESET to cancel the alarm.
  Movement can be resumed after the cancellation. However, the 3D machine interference check will become invalid.
- Inform the machine tool builder if an alarm occurs.

Z82	3D machine interference/Machine model illegal	0002

Details

Machine model is illegal.

Remedy

- Press RESET to cancel the alarm.
  Movement can be resumed after the cancellation. However, the 3D machine interference check will become invalid.
- Inform the machine tool builder if an alarm occurs.

Z82	3D machine Interference check load excess	0003

Details

The calculation of the interference check took time and caused a deceleration.

Remedy

- Inform the machine tool builder.
- Restart the axis in case of a manual operation.
- In case of an automatic operation, the operation will automatically resume when the processing load of the interference check decreases.

Z82	3D machine Interference check error	0004

Details

The interference check failed.

Remedy

- Take a note of the failed status and contact the service center.
- Press RESET to cancel the alarm. Invalidate the 3D machine interference check to continue the operation.

Z84	Unable to save all the history	

Details

The system is unable to store the data due to lack of free space on the internal memory.

Remedy

- Ensure sufficient free space on the internal memory.

Z85	OP panel I/O not connected	0001

Details

No operation panel I/O is connected.

Remedy

- Make sure that an operation panel I/O unit is connected.
- Make sure that power is supplied to the operation panel I/O unit.
- Check for any cable disconnection between the CNC unit and operation panel I/O unit.

(*) If you connect no operation panel I/O unit to the CNC unit, set the parameter "#1261 set33/bit1" to "1".

Z85	No display conn. to panel I/O	0002

Details

No display unit is connected to the operation panel I/O unit.

Remedy

- Make sure an operation panel I/O unit for M800VW/M80VW Series is connected.
- Make sure the display unit is being powered.
- Check for any cable disconnection between operation panel I/O and display units.

(*) If operation panel I/O is not connected to display unit in your system configuration, set the parameter "#1261 set33/bit2" to "0".

Z85	Power ON sequence error	0003

Details

Power ON sequence has not been executed correctly.

Remedy

- Make sure an operation panel I/O unit for M800VW/M80VW Series is connected.
- Make sure the operation panel I/O unit is being powered.
- Check the power-off wiring of the display unit.
- Check for any cable disconnection between CNC and operation panel I/O units.
- Make sure the operation panel I/O and display units are connected.

(*) If no operation panel I/O is connected in your system configuration, set the parameter "#1261 set33/bit1" to "1" and "#1261 set33/bit2" to "0".

(*) If operation panel I/O is not connected to display unit in your system configuration, set the parameter "#1261 set33/bit2" to "0".

Z85	Power OFF sequence error	0004

Details

Power OFF sequence has not been executed correctly.

Remedy

- Make sure an operation panel I/O unit for M800VW/M80VW Series is connected.
- Make sure the operation panel I/O unit is being powered.
- Check the power-off wiring of the display unit.
- Check for any cable disconnection between CNC and operation panel I/O units.
- Make sure the operation panel I/O and display units are connected.

(*) If no operation panel I/O is connected in your system configuration, set the parameter "#1261 set33/bit1" to "1" and "#1261 set33/bit2" to "0".

(*) If operation panel I/O is not connected to display unit in your system configuration, set the parameter "#1261 set33/bit2" to "0".

Z85	Display unit shutoff timeout	0005

Details

Timeout has occurred during wait for the display power shutdown when automatic power OFF is being executed.

Remedy

- Make sure an operation panel I/O unit for M800VW/M80VW Series is connected.
- Make sure the operation panel I/O unit is being powered.
- Check for any cable disconnection between CNC and operation panel I/O units.
- Make sure the operation panel I/O and display units are connected.

(*) If no operation panel I/O is connected in your system configuration, set the parameter "#1261 set33/bit1" to "1" and "#1261 set33/bit2" to "0".

(*) If operation panel I/O is not connected to display unit in your system configuration, set the parameter "#1261 set33/bit2" to "0".

Z86	Overvoltage detection warning	0001

Details

Overvoltage was detected in hardware voltage.

Remedy

- If this alarm occurs frequently, check the power supply voltage.

Z86	Overvoltage detection alarm	0002

Details

Overvoltage was detected in hardware voltage.

Remedy

- Turn OFF and ON the NC power.
- If this alarm occurs frequently, check the power supply voltage.

Z92	Memory ECC error	0004

Details

Incorrect data has been read out from the internal memory.

Remedy

- Contact the service center.

Z95	Image input ex. card sync lost	0001

Details

Synchronization of the image input signal has been lost.

Remedy

- Check the HDMI cable for any disconnection.
- If synchronization loss occurs frequently, exchange the image input expansion card.

Z95	Image input ex. card illegal	0002

Details

The image input expansion card for M80/M800S is installed.

Remedy

- Install the image input expansion card for M80V/M800VS.

Z95	Image input ex. slot illegal	0003

Details

The image input expansion card is installed in the expansion slot EXT2.

Remedy

- Install the image input expansion card in the expansion slot EXT1.

Z96	Power OFF prohibit process ON	0001

Details

Power OFF prohibition process is in execution.

0001: Machine tool builder macro program is being saved.

Remedy

- Do not turn OFF the power while the message is displayed.

Z101	TOOL MNG DATA mismatched	0001

Details

There is a mismatch between the tool life data and the tool management data.

After the tool life data is determined, the number of tools to manage the life has been updated by:

- Changing the value of the parameter "#1439 Tlife-SysAssign" (Part system allocation of life management tools) or "#12055 Tol-lifenum" (Number of life management tools for allocation);
- Changing either the "#1001 SYS_ON" (System validation setup) parameter or the "life management tools" option setting while setting the "#1439 Tlife-SysAssign" (Part system allocation of life management tools) parameter to "0".

Remedy

- Try [Erase tools] from the maintenance screen, and then restart the CNC. Set the tool life data again.

(*) Note that this will also clear the tool compensation data.
Make sure that the tool compensation data is saved with the tool offset data file (TOOL.OFS) before pressing [Erase tools].

Z102	FL-net parameter error	(Parameter No.)

Details

The setting of #[Parameter No.] is incorrect.

Remedy

- Correct the setting.
- For details, refer to the list of messages in "FL-net specifications manual".

Z103	FL-net communication error	(Error code)

Details

A communication error has occurred in FL-net communication.

Error code: indicates the status of CNC (FL-net) (in hexadecimal)

Remedy

- For details of the alarm number, refer to the list of messages in "FL-net specifications manual".

Z104	FL-net node warning	(Node No.)	(Error code)

Details

An error has occurred on the other node that is participating in FL-net.

Node No.: indicates the number of the node in error (in decimal with three digits)

Error code: indicates the status of the node in error (in hexadecimal)

Remedy

- For details of the alarm number, refer to the list of messages in "FL-net specifications manual".

Z105	End-user parameter error

Details

The value of end-user parameter is incorrect.

The value is inconsistent with the multi-project parameter of the built-in PLC, or the sum of the initial device number and the quantity of the devices has exceeded the upper limit.

Remedy

- Correct the value of the end-user parameter displayed.

Z106	VCC card authentication error	0001

Details

Failed to authenticate the vibration cutting expansion unit.

Remedy

- Contact the service center.
- Press the reset to cancel the warning and continue the operations.

Z107	NC-HPU optical comm. warning	0001

Details

A communication error occurred in the NC-HPU optical communication while parameter "#1762 cfgPR12/bit1" is set to "0".

bit0: Communication start error

bit1: Initial communication error

bit2: Communication error

Remedy

- Check that the cable is properly connected.
- Check that the HPU power is turned ON.

Z107	NC-HPU optical comm. alarm	0002

Details

A communication error occurred in the NC-HPU optical communication while parameter "#1762 cfgPR12/bit1" is set to "1".

bit0: Communication start error

bit1: Initial communication error

bit2: Communication error

Remedy

- Check that the cable is properly connected.
- Check that the HPU power is turned ON.
- Turn OFF/ON the NC power.

Z108	HPU error	0001

Details

An error has arisen in the HPU.

bit0: E0 System/HW/SD/optical communication error

bit1: E1 RIO/AI/DI/SIO error

bit2: E2 Network error

bit3: E3 Analyzing process error

bit4: E4 (spare)

bit5: E5 (spare)

bit6: E6 User customization

bit7: E7 User customization

Remedy

- Check the HPU status.

Z109	HPU preparation error	

Details

Registration and deletion of the HPU custom module has been failed.

[Error details]

0001: The total size of the HPU custom module exceeded 5 MB.

0002: Failed to load the HPU custom module.

0003: Failed to load the HPU custom module in the HPU.

0004: The HPU custom module was transferred and deleted while the HPU custom module is already loaded.

0005: The HPU custom module to be deleted did not exist in the HPU.

0006: An error has occurred with the SD card in the HPU.

0020 to 00FF: An internal error has occurred.

Remedy

Take the following procedures depending on the details of the error.

0001: Correct the HPU custom module.

0002: Re-register the HPU custom module.

0003: Correct the HPU custom module.

0004: Restart the NC and HPU.

0005: Initialize the MTB password of the HPU.
Re-register the HPU custom module.

0006: Contact the service center.

0020 to 00FF: Restart the NC and HPU.

If the problem persists, contact the service center.

Z111	Laser I/F unit not connected	0001

Details

The laser I/F unit FCU8-DX522 is not connected.

Remedy

- Connect the laser I/F unit FCU8-DX522, and then reboot the NC.

Z111	Laser I/F unit has no software	0002

Details

There is no software on the laser I/F unit FCU8-DX522.

Remedy

- Set the type of laser oscillator to be connected in parameter "#90001 LASER_UNIT_TYPE", and then reboot the NC.
- Set "1" in parameter "#90002 LASER_UNIT_WRITE", and then reboot the NC.

Z111	Laser I/F unit S/W mismatch	0003

Details

The software of the laser I/F unit FCU8-DX522 does not match parameter "#90001 LASER_UNIT_TYPE".

Remedy

- After setting the type of the laser oscillator to be connected in parameter "#90001 LASER_UNIT_TYPE", set "1" in parameter "#90002 LASER_UNIT_WRITE", and reboot the NC.

Z111	Laser I/F unit H/W error	0004

Details

An error was detected on the laser I/F unit FCU8-DX522.

Remedy

- Write the firmware to the laser I/F unit.
- A failure of the firmware or hardware is suspected.
  Contact the service center.

Z111	No spec: Laser processing ctrl	0005

Details

Laser processing control is not included in the specifications.

Remedy

•Check the specifications.

Z113	MCR init process error

Details

The initialization of motion control release was not processed correctly.

Remedy

•Report this error to the machine tool builder.

Z115	HPU I/F unsupported

Details

The NC, the high speed processing unit or the edge PC does not support the high speed processing unit I/F.

[n1: Displays the function for which the request failed. (DEC)]

2: HPU custom module 1

3: HPU custom module 2

4: HPU custom module 3

5: HPU custom module 4

6: HPU custom module 5

[n2: Displays the system that does not support the high speed processing unit I/F. (DEC)]

0: NC

1: High speed processing unit

2: Edge PC

Remedy

•Upgrade the system.

Z116	HPU I/F request failed

Details

An error occurred in the edge PC for the request by the high speed processing unit I/F.

[n1: Displays the function for which the request failed. (DEC)]

2: HPU custom module 1

3: HPU custom module 2

4: HPU custom module 3

5: HPU custom module 4

6: HPU custom module 5

[n2: Displays the error number detected in the RT Processing unit or the edge PC. (DEC)]

If the error number is between 1 and 9999, look it up in the relevant errors in "IoT Unit Edge PC App Specification manual (BNP-C3087-008)".

If the error number is between 10001 and 19999, look it up in the manual issued by the machine tool builder.

Remedy

•Check the state of the RT Processing unit or the edge PC.

Z117	Ext com var aloc prmtr invalid

Details

The parameter for allocating extended common variables is incorrect.

Remedy

•Correct the parameter that designates allocation of the extended common variables.
When this alarm is caused by more than one parameter, the last parameter number is indicated.

Z118	Can't create ext com var file	

Details

Creating the extended common variable data file failed.

Remedy

- Ensure that sufficient space is left on the SD card in the control unit (M800VW/M80VW) or on the SD card inserted in the SD card slot on the back of the display unit (M800VS/M80V).
- Set the parameter "#11829 exComVar3FileN" to be the same as or smaller than [the number of creatable files].

Z119	12V power error	Error code

Details

NC unit detected low 12V power voltage.

Error code: 24V power voltage measurement value (V)

Remedy

- Cancel the alarm by resetting.
- If this alarm occurs frequently, the hardware may be damaged. Contact our service center.

Z120	Cloud communication error	

Details

Communication with the cloud server failed.

Remedy

- Correct the remote service-related parameters.
- Set "0" in parameter "#8170 Remote Service".

Z121	Dign. report config file error	

Details

The contents of the diagnosis report configuration file are incorrect.

Remedy

- Correct the contents of the diagnosis report configuration file.
- Set "0" in parameter "#8170 Remote Service".
- Set "0" in parameter "#8171 Diag Report Enable".

Z123	CCIETSN (remote) comm. error	n1

Details

A communication error has occurred in the communication that uses a CC-Link IE TSN expansion unit.

n1: indicates the alarm No. (in hexadecimal notation).

Alarm No. indicates the details of the communication error.

Refer to the list of messages in "CC-Link IE TSN (remote station) specification manual".

Remedy

- Take measures described in the list of messages in "CC-Link IE TSN (remote station) specification manual".

Z124	CCIETSN (remote) param. error	Parameter No.

Details

The parameter setting is incorrect.

Parameter setting of #[Parameter No.] is incorrect.

Refer to the list of messages in "CC-Link IE TSN (remote station) specification manual".

Remedy

- Take measures described in the list of messages in "CC-Link IE TSN (remote station) specification manual".

Z125	WLAN_DHCP server error	0001

Details

The DHCP server setting is invalid.

- A value larger than 254 is assigned to the 4th octet of the IP addresses provided by the DHCP server. (The 4th octet is the last portion in an IP address separated by a dot.)

Remedy

- Correct the setting of "#75052 StartIP DHCPserver" or "#75053 IP num DHCPserver" so that a number larger than 254 is not included in the 4th octet of the IP addresses provided by the DHCP server.

Z125	WLAN_DHCP IP address invalid	0002

Details

The DHCP server setting is invalid.

- The IP address of the NC unit and the IP address provided by the DHCP are not in the same network.
- "#75052 StartIP DHCPserver" is set to "0.0.0.0" or "255.255.255.255".

Remedy

- Correct the setting of "#75001 WLAN IP address" or "#75052 StartIP DHCPserver".

Z125	WLAN_SSID not set	0003

Details

The connect SSID or SSID is not entered.

Remedy

- Set "#75003 connect SSID" or "#75005 SSID".

Z125	WLAN_Encryption key not set	0004

Details

The encryption key is not entered.

Remedy

- Set "#75007 Encryption key".

Z125	WLAN_STA/AP mode invalid	0005

Details

The STA/AP mode setting is invalid.

Remedy

- Correct the setting of "#75000 STA/AP mode".

Z125	WLAN_Frequency band invalid	0006

Details

The frequency band setting is invalid.

Remedy

- Correct the setting of "#75009 Frequency band".

Z125	WLAN_Encryption mode invalid	0007

Details

The encryption mode setting is invalid.

Remedy

- Correct the setting of "#75006 Encryption mode".

Z125	WLAN_Channel invalid	0014

Details

The wireless LAN channel setting is invalid.

- The channel you set cannot be used in the country or region corresponding to the country/region code.
- The frequency band corresponding to "#75008 WLAN channel" does not match "#75009 Frequency band".

Remedy

- Correct the setting of "#75008 WLAN channel" to match the country/region code.
- Correct the setting of "#75008 WLAN channel" or "#75009 Frequency band".

Z125	WLAN_C/R Code invalid	0015

Details

The country/region code setting is invalid.

- Wireless LAN has not been approved in the country or region corresponding to the country/region code.
- A nonexistent country/region code is set.
- No country/region code is entered.

Remedy

- Reboot the NC. If the alarm continues to occur after reboot, have the machine tool builder set the country/region code, and then reboot the NC.
- Set "#75000 STA/AP mode" to "0" to disable wireless LAN.

Z125	WLAN_Setting illegal	0050

Details

There is an error in the wireless LAN setting.

Remedy

- Correct the setting of wireless LAN.

Z125	WLAN_Startup error	0051

Details

Wireless LAN failed to start.

Remedy

- Reboot the NC.
- If the alarm continues to occur after reboot, contact your local service center.

Z125	WLAN_Internal error	0x00C8 to 0x00FF

Details

An internal error occurred in wireless LAN.

Remedy

- Contact the service center.
- Set "#75000 STA/AP mode" to "0" to disable wireless LAN.

Z128	ROM write error

Details

- The machine tool builder macro program has not been successfully written to ROM.
- Formatting of the machine tool builder macro program area has not been successfully completed.

[Type]

0001: Open error

0002: Erase error

0003: Write error

0004: Verify error

Remedy

- Write the program to ROM again.
- Format the area again.

(*) If the alarm is not cleared, the hardware may be broken. Contact the service center.

Z130	MTB macro file invalid

Details

- The machine tool builder macro file was entered into an NC that has a different passcode than the NC from which the file was output.
- You tried to enter a machine tool builder macro file that is larger than the storage capacity.

Remedy

- Verify the passcode and size, and enter the machine tool builder macro file again.

Z131	OMR-CC dr unit not spt

Details

A drive unit that does not support OMR-CC is connected to the axis with "#2678 OMRCC_valid" set to "1".

Remedy

- To enable OMR-CC, the drive units for all axes with "1" set in "#2678 OMRCC_valid" must be OMR-CC compatible MDS-E series version B3 or later.
- To disable OMR-CC, set "#2678 OMRCC_valid" to "0".

6

Absolute Position Detection System Alarms (Z7*)

Z70	**Abs posn base set incomplete**	0001	(Axis name)

Details

Zero point initialization is incomplete. Otherwise, the spindle was removed.

Remedy

Complete zero point initialization.

Z70	**Absolute position lost**	0002	(Axis name)

Details

[Absolute position lost]

The absolute position basic point data saved in the NC has been damaged.

[Absolute position lost due to replacement of encoder]

The batteryless encoder has been replaced.

Remedy

[Absolute position lost]

Set the parameters. If the basic point data is not restored by setting the parameters, perform zero point initialization.

[Absolute position lost due to replacement of encoder]

Initialize the zero point.

Z70	**Abs posn param changed**	0003	(Axis name)

Details

Any of the parameters for absolute position detection has been changed.
#1003 iunit
#1017 rot
#1018 ccw
#1040 M_inch
#2049 type
#2201 PC1
#2202 PC2
#2218 PIT
#2219 RNG1
#2220 RNG2
#2225 MTYP

Remedy

Correct the parameter settings. Then turn the power ON again and perform zero point initialization.

Z70	**Abs posn initial set illegal**	0004	(Axis name)

Details

The zero point initialization point is not at the grid position.

Remedy

Perform the zero point initialization again.

Z70	**Abs posn param restored**	0005	(Axis name)

Details

The data has been restored by inputting the parameters during the alarm No.0001, 0002, and 0003.

Remedy

Turn the power ON again to start the operation.

Z70	**Abs data error**	0006

Details

Deviation of the servo axis with scale when the power is OFF exceeds the set value in "#2051 check" (Check).

Remedy

Search for the factor which led the deviation of the servo axis at the power OFF.

Z70	**Abs posn data lost**	0080	(Axis name)

Details

The absolute position data has been lost. An error of the multi-rotation counter data in the encoder and so on may be the cause. (Liquid penetrates into encoder connector, etc.)

Remedy

Replace the encoder and complete zero point initialization.

Z70	**Abs posn error(servo alm 25)**	0101	(Axis name)

Details

The servo alarm No. 25 was displayed and the power was turned ON again.

Remedy

Perform zero point initialization again.

Z70	**Abs posn error(servo alm E3)**	0106	(Axis name)

Details

The servo alarm No. E3 was displayed and the power was turned ON again.

Remedy

Perform zero point initialization again.

Z71	**AbsEncoder:Backup voltage drop**	0001	(Axis name)

Details

Backup voltage in the absolute encoder dropped.

Remedy

Replace the battery, check the cable connections, and check the encoder. Turn the power ON again and perform zero point initialization.

Z71	**AbsEncoder: Commu error**	0003	(Axis name)

Details

Communication with the absolute encoder has been disabled.

Remedy

Check and replace the cables, card or encoder. Turn the power ON again and perform zero point initialization.

Z71	**AbsEncoder: Abs data changed**	0004	(Axis name)

Details

Absolute position data has been changed at the absolute position establishment.

Remedy

Check and replace the cables, card or encoder. Turn the power ON again and perform zero point initialization.

Z71	**AbsEncoder: Serial data error**	0005	(Axis name)

Details

An error of the serial data was found in the absolute encoder.

Remedy

Check and replace the cables, card or encoder. Turn the power ON again and perform zero point initialization.

Z71	**AbsEncoder: Abs/inc posn diffr**	0006	(Axis name)

Details

Servo alarm E3
Absolute position counter warning

Remedy

Operation is possible until the power is turned OFF.

Z71	**AbsEncoder: Initial commu er**	0007	(Axis name)

Details

Initial communication with the absolute encoder is not possible.

Remedy

Check and replace the cables, card or encoder. Turn the power ON again and perform zero point initialization.

Z72	**Position check error**	(Axis name)

Details

This alarm is displayed if an error is detected when comparing the encoder's absolute position and controller coordinate values in the absolute position system.

Z73	**Battery for abs data fault**	0001

Details

Low backup battery
Servo alarm 9F
Low battery voltage

Remedy

If the battery voltage is low or the cable is damaged, there is no need to initialize the absolute position.

7

Distance-coded Reference Scale Errors (Z8*)

Z80	**Basic position lost**	0001

Details

The basic point data saved in the NC has been damaged.

Remedy

- Set the parameters. If the basic point data is not restored by setting the parameters, perform zero point initialization.

Z80	**Basic position restore**	0002

Details

The basic point data has been restored by setting the parameters.

Remedy

- Turn the power ON again to start the operation.

Z80	**No spec: Distance-coded scale**	0003

Details

The distance-coded reference scale has been set available although this function is out of the specifications.

Remedy

- Check the specifications.
- If you do not use this function, correct the encoder type with the servo parameter.

Z81	**R-pos adjustment data lost**	0001

Details

Reference position adjustment value data saved in the CNC has been damaged.

Remedy

- Set the parameter. If the data is not restored by setting the parameter, establish the reference position again.

Z81	**R-pos adjustment data restored**	0002

Details

After the 'Z81 R-pos adjustment data lost 0001', the data has been recovered by setting the parameter.

Remedy

- Establish the reference position to start the operation.

Z83	**NC started during SP rotation**	0001

Details

The NC was started while the spindle was rotating.

Remedy

- Turn OFF the power once, and make sure that the spindle is stopped before turning ON the power again.
- You can select whether or not to stop the spindle forcibly at the occurrence of this alarm by the setting of the base common parameter "#1284 ext20/bit1" (Spindle control selected in response to Z83 (NC started during SP rotation)).
- If this alarm occurs frequently, check the power supply voltage.

8

Emergency Stop Alarms (EMG)

EMG	Emergency stop	PLC

Details

The "PLC emergency stop" signal is turned ON during the sequence processing of the user PLC.

Remedy

- Check the conditions to turn the "Emergency stop" signal ON and remove the cause of the alarm during the sequence processing of the user PLC.

EMG	Emergency stop	EXIN

Details

The "emergency stop" signal is significant (open).

Remedy

- Cancel the "emergency stop" signal.
- Check for any broken wires.

EMG	Emergency stop	SRV

Details

An alarm occurred in the servo system causing an emergency stop.

Remedy

- Investigate and remove the cause of the servo alarm.

EMG	Emergency stop	STOP

Details

The user PLC (ladder sequence) is not running.

Remedy

- Check the setting of the control unit rotary switch CS2. Correct it if set to "1".
- Check the [RUN/SP] (run/stop) switch on the PLC edit file save screen (onboard function). Turn it OFF if ON.

EMG	Emergency stop	SPIN

Details

Spindle drive unit is not mounted.

Remedy

- Cancel the causes of the other emergency stop.
- Check the "emergency stop" signal input in the spindle drive unit.

EMG	Emergency stop	PC_H

Details

Failure in the high-speed PC processing abnormal

Remedy

- Correct the sequence program. (To stop monitoring the high-speed PC processing temporarily, set "1" in "#1219 aux03/bit1 (Stop high-speed PC monitoring function)". Disable the monitoring function only as a temporary measure.)

EMG	Emergency stop	PARA

Details

Setting of the door open II fixed device is illegal.
Setting of the parameters for dog signal random assignment is illegal.

Remedy

- Correct the "#1155 DOOR_m" and "#1156 DOOR_s" settings. (When the door open II fixed device is not used, set "#1155 DOOR_m" and "#1156 DOOR_s" to "100".)
- Correct the "#2073 zrn_dog (Origin dog Random assignment device)", "#2074 H/W_OT+ (H/W OT+ Random assignment device)", "#2075 H/W_OT- (H/W OT- Random assignment device)" and "#1226 aux10/bit5 (Arbitrary allocation of dog signal)" settings.

EMG	Emergency stop	LINK

Details

An emergency stop occurs when the FROM/TO instruction is not executed within 500ms.

Remedy

- Execute the FROM/TO instruction one or more times every 500ms.

The time in which no interrupt request is issued from MELSEC is measured and stored in the following R registers:
R10190: Current timeout counter
R10191: Maximum timeout counter after power ON
R10192: Maximum timeout counter after system is started up (this is backed up)

Details

MELSEC is in error and reset states.

Remedy

- Check the MELSEC states.

Details

The contents of MELSEC-specific code area in buffer memory have been damaged.

Remedy

- Check the MELSEC states.

Details

PLC serial link communication has stopped.

Remedy

- Check the CC-Link card wiring and the external sequencer transmission.
- Check the link communication errors shown on the diagnostic screen.

EMG	Emergency stop	WAIT

Details

The preparation sequence is not sent from the master station. Otherwise, the contents of the received preparation sequence are inconsistent with those of the parameters, so that the usual sequence cannot be started.

Remedy

- Check that the CC-Link card switch setting and wiring as well as the external sequencer transmission are normal.
- Check the diagnostic screen for link communication errors.

EMG	Emergency stop	XTEN

Details

The CC-Link card is operating incorrectly.
Switch/parameter settings for the CC-Link card are incorrect.

Remedy

- Replace the CC-Link card.
- Correct the switch/parameter settings for the CC-Link card.

EMG	Emergency stop	LAD

Details

The sequence program has an illegal code.

Remedy

- Correct any illegal device Nos. or constants in the sequence program.

EMG	Emergency stop	CVIN

Details

The "emergency stop" signal for power supply is significant (open) because the external emergency stop function for power supply is enabled.

Remedy

- Cancel the "emergency stop" signal.
- Check for any broken wires.

EMG	Emergency stop	MCT

Details

The contactor shutoff test is being executed.

Remedy

- The emergency stop is reset automatically after the contactor shutoff is confirmed.
- If the contactor shutoff is not confirmed within 5 seconds after the "contactor shutoff test" signal has been input, the "contactor welding detected" alarm occurs and the emergency stop status remains.
- Make sure that the contactor's auxiliary b contact signal is correctly output to the device that is set in "#1330 MC_dp1" and "#1331 MC_dp2" (Contactor weld detection device 1 and 2), and then turn the power ON again.

EMG	Emergency stop	IPWD

Details

The data backup for power failure might not have been executed successfully at the previous power failure.

Remedy

- If this message appears frequently, the power supply may be deteriorated. Contact the service center.

EMG	Emergency stop	ENC

Details

The encoder replacement operation is in progress.

Remedy

- Complete the encoder replacement operation on the "ABS. POSITION SET" screen.

EMG	Emergency stop	LINE

Details

- Emergency stop "LINE" was retained due to a momentary OFF of the external emergency stop signal.
- The emergency stop signal of the drive unit was detected while Ready ON was active on the NC.
- The emergency stop signal of the drive unit cannot be canceled.

Remedy

- Check the alarm history. If Emergency stop "EXIN" was issued immediately prior to Emergency stop "LINE", check the wiring of the external emergency stop. Emergency stop "LINE" can be canceled by activating the external emergency stop.
- Check the connection of the drive unit.

9

Computer Link Errors (L)

L01	Timeout error	0004

Details

Communication ended with timeout.
(CNC has a 248-byte receive buffer. The time during which CNC receives 248 bytes exceeds the "TIME-OUT" value set in the I/O device parameter.)

Remedy

- Set a greater timeout value in the input/output device parameter.
- Check the software in HOST and make sure that the HOST transmits data in response to DC1 (data request) from CNC.
- Set "#9614 START CODE" to "0".

L01	Host ER signal OFF	0010

Details

ER signal in HOST (or DR signal in CNC) is not turned ON.

Remedy

- Check for any disconnected cable.
- Check for any broke wire.
- Make sure that the HOST power is turned ON.

L01	Parity H error	0015

Details

Communication ended with parity H.

Remedy

- Check the software in HOST and make sure that the data to be transmitted to CNC is ISO code.

L01	Parity V error	0016

Details

Communication ended with parity V.

Remedy

- Correct the data to transmit to CNC.

L01	Overrun error	0017

Details

CNC received 10 bytes or more data from HOST in spite of DC3 (request to stop data transfer) transmission from CNC to the HOST, which terminated the communication.
CNC received 10 bytes or more data from HOST during the data transmission from CNC to the HOST.

Remedy

- Check the software in HOST and make sure that the HOST stops transmitting data within 10 bytes after receiving DC3.
- Correct the software in HOST not to transmit data such as a command or header to CNC during receiving a machining program.

10

User PLC Alarms (U)

(Note) U10 Illegal PLC (User PLC is illegal)

- "xx" in the lower 16 bits of the sub-status 1 indicates the program No. (0x01 to 0x78(in the hexadecimal form))
- The sub-alarm No. "yy" of sub- status 1 indicates the project No. ("yy" is not displayed when the maximum number of projects is 1)

(Note) For details of user PLC alarms, refer to "PLC Development Manual".

U01	**No user PLC**	-	-

Details

No sequence program is included in the built-in ROM or temporary memory area.

(1) Sequence program is not stored in the built-in ROM.

(2) Sequence program is not written from the GX Developer/GX Works2 or internal PLC edit function.

(3) Sequence program cannot be read due to broken built-in ROM.

(4) No large capacity PLC additional specifications.

Remedy

(1) Write the sequence program from the GX Developer/GX Works2 or internal PLC edit function, and then execute the built-in ROM writing.

(2) Check the presence of additional specification "Large PLC capacity" and reconsider the storable size.

(3) If (1) or (2) does not solve the problem, there is a possibility that built-in ROM is broken.

U10	**Illegal PLC**	0x04xx.yy	Step No.

Details

Software instruction interruption illegal

An error was found in data for the sequence program in execution.

(1) Sequence program stored in the built-in ROM is broken.

(2) Sequence program under development (before writing into F-ROM) is broken.

Remedy

Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x100*	-

Details

A H/W error was detected during the PLC execution.

Remedy

Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x110*	-

Details

The PLC system execution preparation failed.

Remedy

Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x120*	-

Details

Number of ladder over (at PLC system startup)

The total number of "ladder files" stored in built-in ROM exceeded the maximum.

Remedy

Write "sequence program", "comment file", "PLC message file", and "symbolic information file" again.

U10	Illegal PLC	0x130*	-

Details

Number of data over (at PLC system startup)

The total number of "comment files", "PLC message files", and "symbolic information files" stored in built-in ROM exceeded the maximum.

Remedy

Remedy the error in either of the following methods.

(1) Select [Format PLC memory] with GX Developer/GX Works2 and format the memory after setting [Target memory] to [Memory card(RAM)]. After that, write "comment file", "PLC message file", and "symbolic information file" again.

(2) Create a ladder with [Add New Data] in built-in PLC editing function and execute "Format". After that, write "sequence program", "comment file", "PLC message file", and "symbolic information file" again.

U10	Illegal PLC	0x20xx.yy	Step No.

Details

Label branching error (Before executing PLC)

Occurs only when the bit selection parameter (#6452 bit6) "branch destination label check valid" is set to "1".

(1) The CJ and CALL instructions were placed to a nonexistent label.

(2) The CJ instruction was placed to the global label. (Branching is possible only with the CALL instruction.)

Remedy

Check the branch destination of the CJ and CALL instructions existing in the steps occurred.

U10	Illegal PLC	0x21xx.yy	Step No.

Details

Label duplication error (Before executing PLC)

(1) When using the multi-programming method:
- Labels of common pointer are duplicated.
- Labels of local pointer are duplicated within the same file.

(2) When using the independent program method, labels are duplicated.

Remedy

Correct the duplication of the labels existing in the steps occurred.

U10	Illegal PLC	0x22xx.yy	-

Details

Local label over (Before executing PLC)

(*) "Local label" is as "labels of local pointer".

The boundary value set with the PC parameter (common pointer boundary value) has been exceeded by the total number of labels of local pointer.

Remedy

(1) Reduce the number of local labels used.
- Use as sequentially as possible from P0.

(2) Reset the PC parameter (common pointer boundary value).

U10	Illegal PLC	0x230*.yy	-

Details

Global label boundary value error (Before executing PLC)

(*) "Global label" is as "labels of common pointer".

The content of PC parameter (common pointer boundary value) is not normal.

(1) When using the multi-programming method, a value greater than the maximum value is set.

(2) When using the independent program method, the label boundary value of common pointer is set.

Remedy

(1) When using the multi-programming method, correct the common pointer boundary value to an appropriate value.

(2) When using the independent program method, delete the common pointer boundary value.

U10	Illegal PLC	0x24xx.yy	Step No.

Details

Reserved label error (Before executing PLC)

(1) When using the multi-programming method, disabled reserved label exists.

(2) When using the independent program method, reserved labels are duplicated.

Remedy

(1) When using the multi-programming method, delete the reserved label.

(2) When using the independent program method:
- Delete the PC parameter program settings.
- Correct the duplication of reserved labels.

U10	Illegal PLC	0x25xx.yy	-

Details

Program setting error (Before executing PLC)

(1) When using the multi-programming method, PC parameter setting is not correct.
- PC parameter (program setting) is not set.
- Unstored program name is set.
- The contents of the program name is abnormal.
- More than the maximum number of programs that can be set (120 programs) are set.

(2) When using the independent program method, multiple programs are stored.

(3) When the multi-project is valid, the sum of all projects exceeds the number of the programs which can be set (120 programs).

Remedy

(1) When using the multi-programming method, check the PC parameter program settings.
- Check the program settings.
- Check the program name stored in the NC.
- Review the program name and rename it if necessary.
  e.g. The program name and the M device number in the PLC program are overlapped.
- Set the number to 120 or less.

(2) When using the independent program method:
- Store only one program file.

(3) When the multi-project is valid, set the number of programs that can be set to be less than 120 programs as the sum of all projects.

U10	Illegal PLC	0x26xx.yy	-

Details

RET instruction error

(1) RET instruction was not executed at the branch destination of the CALL instruction.

(2) RET instruction was executed without execution of CALL instruction.

Remedy

Check the following matters for the entire sequence program to be executed.

(1) Check if RET instruction is programmed at the end of sub-routine.

(2) Check if diverged to the other operation in the middle of sub-routine and RET instruction is not executed.

(3) Check if jumped to the END reservation label (P4005) in the middle of sub-routine.

(4) Check if there is delimiter (FEND instruction) between adjacent program and sub-routine program.

U10	Illegal PLC	0x27xx.yy	Step No.

Details

Ladder code error (Before executing PLC)

An error was found in data for the sequence program to be executed.

(1) Disabled PLC instruction is used.

(2) Devices/labels are used for indexes in array labels.

(3) Sequence program stored in the built-in ROM is broken.

(4) Sequence program under development (before writing into the built-in ROM) is broken.

Remedy

(1) Check the sequence programs and delete disabled PLC commands.

(2) When devices/labels are used for indexes in array labels, set "#4671/bit1" to "1".

(3)(4) Transferring, storing and ROM writing of the sequence program must be re-executed with the GX Developer or PLC onboard edit function.

U10	Illegal PLC	0x280*	-

Details

No main processing ladders (Before executing PLC)

Main processing program to be executed cannot be identified.

(1) When using the multi-programming method, main processing "scan" is not set in the PC parameter (program setting).

(2) When using the independent program method, no reservation ladder for the main processing ladder is available.

Remedy

(1) When using the multi-programming method, check the PC parameter program settings.

(2) When using the independent program method, add the reservation label (P4002) for the medium speed ladder.

U10	Illegal PLC	0x29xx.yy	-

Details

Execution area over (Before executing PLC)

The total number of steps for the ladder to be executed has exceeded the size of PLC processor execution area.

Remedy

Check the PC parameter (program setting) and set so that the total number of steps for the ladder to be executed does not exceed the PLC processor execution area.

U10	Illegal PLC	0x30xx.yy	Step No.

Details

FOR instruction nesting over

17th level of nesting for FOR instruction was executed.

Remedy

Check the number of FOR instruction's nesting depth in the steps to which the error occurred, and limit the number to 16 or less.

U10	Illegal PLC	0x31xx.yy	Step No.

Details

NEXT instruction error

(1) NEXT instruction was executed before FOR instruction.

(2) After FOR instruction, END (FEND) was executed before NEXT instruction.

Remedy

(1) Check and correct the NEXT instruction in the step to which the error occurred.

(2) Check and correct the ladder circuit of the program to which the error occurred. (Note that the step No. at the error position is displayed as "0".)
- Check if JMP,CALL,CJ instructions were executed between FOR and NEXT instruction, and NEXT instruction was jumped.
- Check if FOR instruction and NEXT instruction are all paired.

U10	Illegal PLC	0x32xx.yy	Step No.

Details

BREAK instruction error

BREAK was executed outside the range between FOR and NEXT instruction.

Remedy

Check and correct the BREAK instruction in the step to which the error occurred.

U10	Illegal PLC	0x33xx.yy	-

Details

Structured ladder error
When sequence programs created using structured project are executed.

Remedy

Check if the target sequence program is not created using structured project.

U10	Illegal PLC	0x34xx.yy	Step No.

Details

Data command range error
Description of specified device exceeds command range set on each instruction.

Remedy

Check and correct the usage of instructions existing in the steps occurred.

U10	Illegal PLC	0x35xx.yy	Step No.

Details

Real number calculation data error
When descriptions of specified device are the following values.

♦Outside the range of the following

[single float] 2 to the power of -126 ≤ Specified device < 2 to the power of 128

[double float] 2 to the power of -1022 ≤ Specified device < 2 to the power of 1024

♦-0, Denormalized number, Non-number, ± ∞

Remedy

Check and correct the usage of instructions existing in the steps occurred.

U10	Illegal PLC	0x36xx.yy	Step No.

Details

Real number calculation data error
When the real number calculation result exceeds the following range.

[single float] Calculation result < 2 to the power of 128

[double float] Calculation result < 2 to the power of 1024

Remedy

Check and correct the usage of instructions existing in the steps occurred.

U10	Illegal PLC	0x400*	-

Details

PLC system error

Remedy

Contact Mitsubishi Electric.

U10	Illegal PLC	0x500*	-

Details

Maximum project No. illegal (at PLC system startup)

Multi-project parameter setting is illegal (due to the following reason).

- The value outside the setting range (1 to number of usable project) was detected.

Remedy

(1) Check the presence of the expansion project additional specification and reconsider the setting range.

(2) The multi-project parameter (maximum project No.) setting is illegal. Set the multi-project parameter again.

(3) If (1) or (2) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	Illegal PLC	0x510*	-

Details

Project ratio illegal

Multi-project parameter setting is illegal (due to the following reason).

- It was detected that the total of ratios of all projects was outside the range from 0 to 100.

Remedy

(1) The multi-project parameter (project ratio) setting is illegal. Set the multi- project parameter again.

(2) If (1) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	Illegal PLC	0x520*	-

Details

Temporary memory area over (at the PLC system startup)

Multi-project parameter setting is illegal (due to the following reason).

- The ratio to which the area after the project ratio setting is smaller than the size of ladder stored in the temporary memory area was detected.

Remedy

(1) Check the presence of the large-capacity PLC additional specification and reconsider the size that can be stored.

(2) The multi-project parameter (project ratio) setting is illegal. Set the multi- project parameter again.

(3) If (1) or (2) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	Illegal PLC	0x530*	-

Details

Built-in ROM area over (at the PLC system startup)

Multi-project parameter setting is illegal (due to the following reason).

- The ratio to which the area after the project ratio setting is smaller than the size of ladder stored in the built-in ROM area was detected.

Remedy

(1) Check the presence of the large-capacity PLC additional specification and reconsider the size that can be stored.

(2) The multi-project parameter (project ratio) setting is illegal. Set the multi- project parameter again.

(3) If (1) or (2) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x540*	-

Details

Comment area over (at the PLC system startup)

Multi-project parameter setting is illegal (due to the following reason).

- The ratio to which the area after the project ratio setting is smaller than the size of comment and message stored in the built-in ROM area.

Remedy

(1) The multi-project parameter (project ratio) setting is illegal. Set the multi- project parameter again.

(2) If (1) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x550*	-

Details

Execution project illegal (at the PLC system startup)

Multi-project parameter setting is illegal (due to the following reasons).

- The value outside the setting range (ON/OFF) was detected.
- All execution projects are OFF.

Remedy

(1) The multi-project parameter (execution project) setting is illegal. Set the multi-project parameter again.

(2) If (1) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x560*	-

Details

Project execution order illegal (at the PLC system startup)

Multi-project parameter setting is illegal (due to the following reasons).

- The value outside the setting range (1 to 6) was detected.
- The redundant project execution order number was detected.

Remedy

(1) The multi-project parameter (project execution order) setting is illegal. Set the multi-project parameter again.

(2) If (1) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x570*	-

Details

Parameter setting illegal for the number of common device points (at PLC system startup)

Multi-project parameter setting is illegal (due to the following reasons).

- The value outside the setting range (0 to the minimum number of points in all projects) was detected.
- It was detected that the number of points was not a multiple of 16.

Remedy

(1) Check the presence of the expansion project additional specification and reconsider the setting range.

(2) The multi-project parameter setting is illegal. Set the multi-project parameter again.

(3) If (1) or (2) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	Illegal PLC	0x580*.yy	-

Details

Parameter setting illegal for number of device points (at PLC system startup)

The parameter setting for the number of device points is illegal (due to the following reasons).

- The total number of points in one project 29K or more was set.
- The illegal value (-1 point or less, or 61441 points or more) for the number of device points was detected.
- It was detected that the number of device points was not a multiple of 16.
- The number outside the usable number of device points in whole projects was detected.

Remedy

(1) The parameter setting for the number of device points is illegal. Set the parameter of the number of device points again and reboot the system.

(2) If (1) does not solve the problem, there is a possibility that the NC system is broken. Contact Mitsubishi Electric.

U10	Illegal PLC	0x700*.yy	-

Details

Caution; Ladder program writing during RUN is disabled (In independent program method)
With RUN write ON, sequence program is operated in independent program method. Run write is not available with this condition.

Remedy

The ladder program writing (RUN write) can not be used in independent program method.
Disable RUN write, or change the method into multi-program one.

U10	Illegal PLC	0x71xx.yy	Step No.

Details

Caution; Ladder program writing during RUN is disabled (common pointer is used in high-speed processing)
With RUN write is ON, high-speed program with common pointer is running. RUN write is not available with this condition.

Remedy

RUN write is not available when high-speed processing with common pointer is running. Change the high-speed processing program to the one without common pointer.

U10	Illegal PLC	0x720*	-

Details

Caution; Ladder program writing during RUN is disabled (high-speed processing size is exceeding)
With RUN write ON, the execution size of high-speed processing program has exceeded 4000 steps. RUN write is not available with this condition.

Remedy

RUN write is not available when execution size of high-speed processing program exceeds 4000 steps. Edit the high-speed processing program to reduce the execution size to 4000 steps or less.
(When multi-project is valid, edit the program to reduce the execution size to 4000 steps or less in total of all the project.)

U10	Illegal PLC	0x730*.yy	-

Details

Caution; Ladder program writing during RUN is disabled (number of labels in high-speed processing is beyond the capacity)

(*) "Local label" is as "labels of local pointer".

With RUN write ON, the number of labels of local pointer in high-speed processing program has exceeded 256. RUN write is not available with this condition.

Remedy

RUN write is not available when the number of labels of local pointer in high-speed processing program exceeds 256. Edit the high-speed program to reduce the number of labels of local pointer to less than 256.
(When multi-project is valid, edit the program to reduce the number of labels of local pointer to less than 256 per project.)

U10	Illegal PLC	0x80xx.yy	Step No.

Details

Software exceptional interruption (BCD instruction error) has occurred.

With BCD and DBCD instructions, BIN value outside its input range was attempted to be converted into BCD.

Remedy

Check the usage of BCD, DBCD instructions existing in the steps occurred.

U10	Illegal PLC	0x81xx.yy	Step No.

Details

Software exceptional interruption (BIN instruction error) has occurred.

With BIN and DBIN instructions, BCD value outside its input range was attempted to be converted into BIN.

Remedy

Check the usage of BIN, DBIN instructions existing in the steps occurred.

U10	Illegal PLC	0x82xx.yy	Step No.

Details

Software exceptional interruption (Bus error) has occurred.

When the specified device exceeds the range of the device

Remedy

(1) Check and correct the device range that the instruction in the step of which the error occurred accesses.

(2) Contact Mitsubishi Electric.

U10	Illegal PLC	0x83xx.yy	Step No.

Details

Software exceptional interruption (Unmounted instruction error) has occurred.

(1) When the bit selection parameter (#6452 bit6) "branch destination label check valid" is set to "0", jumped to an undefined label.

(2) Sequence program in execution is broken.

Remedy

(1) Set the bit selection parameter (#6452 bit6) "branch destination label check valid" to "1" and check the branching step to the undefined label.

(2) Contact Mitsubishi Electric.

U10	Illegal PLC	0x84xx.yy	Step No.

Details

Software exceptional interruption (Instruction format error) has occurred.

Remedy

Contact Mitsubishi Electric.

U10	Illegal PLC	0x85xx.yy	Step No.

Details

Software exceptional interruption (Instruction bus error) has occurred.

(1) When the bit selection parameter (#6452 bit6) "branch destination label check valid" is set to "0", jumped to an undefined label.

(2) Sequence program in execution is broken.

Remedy

(1) Set the bit selection parameter (#6452 bit6) "branch destination label check valid" to "1" and check the branching step to the undefined label.

(2) Contact Mitsubishi Electric.

U10	Illegal PLC	0x86xx.yy	Step No.

Details

Software exceptional interruption (CALL/RET instruction error) has occurred.

Remedy

Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x87xx.yy	Step No.

Details

Software exceptional interruption (memory area error) has occurred.

Remedy

Contact Mitsubishi Electric.

U10	**Illegal PLC**	0x8Bxx.yy	Step No.

Details

Software exceptional interruption (ASYNC BUS error) has occurred.

Remedy

Contact Mitsubishi Electric.

U50	**PLC stopped**

Details

The ladder is stopped.

Remedy

Run the PLC.

U55	**PLC stopped / is not saved**

Details

The ladder is stopped and the edited sequence program is not stored in the built-in ROM.
(The contents of temporary memory area and sequence program in the built-in ROM are mismatched.)

Remedy

Run the PLC.

Use GX Developer/GX Works2 or the built-in PLC edit function to perform write to the built-in ROM.

U60	**Ladder is not saved**

Details

The edited sequence program is not stored in the built-in ROM.
(The contents of temporary memory area and sequence program in the built-in ROM are mismatched.)

Remedy

Use GX Developer/GX Works2 or the built-in PLC edit function to perform write to the built-in ROM.

11

Network Service Errors (N)

N001 Modem initial error

Details

An error occurred in the modem connection at the power ON.

Remedy

- Check the connection between the NC and modem, connection port and power supply to modem.

N002 Redial over

Details

- The number of redials exceeded due to the dial transmission failure.

Remedy

- Wait a while, and then dial again.

N003 TEL unconnect

Details

- The phone line is not connected.

Remedy

- Check for any disconnection in the modem's phone line.

N004 Net communication error

Details

- An error other than the above occurred during communication.

Remedy

- Note down how the error occurred and contact the service center.

N005 Invalid net communication

Details

- The modem connection port is being used for another function such as input/output.
- The modem connection port settings are incorrect.

Remedy

- Stop using the modem connection port with the other function, and then turn the power ON again.
- Correct the settings of the modem connection port.

N006 Received result of diagnosis

Details

- A diagnosis data file has been received.

Remedy

- Clear the message.

N007 Send data size over

Details

- A file larger than Anshin-net server capacity (64Kbyte) has been transmitted in machining data sharing.

Remedy

- Reduce the size of machining program file so that it won't exceed the capacity of Anshin-net server.

N008 No file on server

Details

- The file reception failed in machining data sharing because no file exists on Anshin-net server.

Remedy

- Confirm that a machining program file exists on Anshin-net server before receiving it.

N009 Password error

Details

- The file reception failed in machining data sharing due to a wrong password.

Remedy

- Input the password again.

N010 Customer number error

Details

- The file reception failed in machining data sharing due to a wrong customer number.

Remedy

- Input the customer number again.

N011 Storage capacity over

Details

- The file reception failed in machining data sharing because the size of the file to be received is bigger than free space in the NC.

Remedy

- Ensure sufficient free space in the NC.

N012 File deletion error

Details

- A file on Anshin-net server cannot be deleted in machining data sharing.

Remedy

- Confirm that the file exists on Anshin-net server.
- Note down how the error occurred and contact the service center.

12

Program Errors (P)

These alarms occur during automatic operation, and the causes of these alarms are mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

P10 No. of simultaneous axes over

Details

The number of axis addresses programmed in a block exceeds the specified maximum.

Remedy

- Divide the command programmed in the block into two blocks.
- Check the specifications.

P11 Illegal axis address

Details

The axis address commanded by the program does not match any of the ones set by the parameter.

Remedy

- Correct the axis names in the program.

P20 Division error

Details

- The issued axis command cannot be divided by the command unit.
- A cutting feed command has been issued to the index table indexing axis when the parameter "#2580 index_ Gcmd" is set to "1".

Remedy

- Correct the program.

P29 Not accept command

Details

The command has been issued when it is impossible.

- The normal line control command (G40.1, G41.1, G42.1) has been issued during the modal in which the normal line control is not acceptable.
- The command has been issued during the modal in which the two-part system simultaneous thread cutting is not acceptable.
- A workpiece coordinate system preset command (G92.1) has been issued during tool length compensation, nose R compensation or tool radius compensation.
- Thread groove position command (G123) has been given under any of the following conditions.
  - Lead axis coordinates have not been determined
  - Phase Z of the spindle has not been detected
  - Coordinate rotation by program is active
- Address C of the thread groove position command (G123) has been specified under any of the following conditions.
  - C-axis coordinates have not been determined
  - Any mode other than C-axis mode is active

Remedy

- Correct the program.
- Establish the following conditions before issuing the thread groove position command (G123).
  - Lead axis coordinates have been determined
  - Phase Z of the spindle has been detected
  - Coordinate rotation by program is not active
- Establish the following conditions before specifying address C of the thread groove position command (G123).
  - C-axis coordinates have been determined
  - C-axis mode is active

P29	Not accept command	4

Details

- Coordinate system setting command (G92) has been given while tool length offset is temporarily canceled by G53, G28, G30 or G30.1 command. (Note)
- You changed tool length offset with tool length compensation enabled (G43/G44) and then issued a coordinate system setting command (G92) before the offset was reflected.

(Note) The alarm does not occur if G92 and G53 are put in a block (G92 G53 X0 Y0 Z0;).

Remedy

- Correct the program so that the coordinate system setting command (G92) is given after the tool length offset is properly reflected.

P30	Parity H error

Details

The number of holes per character on the paper tape is even for EIA code and odd for ISO code.

Remedy

- Check the paper tape.
- Check the tape puncher and tape reader.

P31	Parity V error

Details

The number of characters per block on the paper tape is odd.

Remedy

- Make the number of characters per block on the paper tape even.
- Set the parameter parity V selection OFF.

P32	Illegal address

Details

An address not listed in the specifications has been used.

Remedy

- Correct the program address.
- Correct the parameter settings.
- Check the specifications.

P33	Format error

Details

The command format in the program is not correct.

Remedy

- Correct the program.

P34	Illegal G code

Details

- The commanded G code is not in the specifications.
  An illegal G code was commanded during the coordinate rotation command.
- G51.2 or G50.2 was commanded when the rotary tool axis No. (the parameter "#1501polyax") was set to "0".
  G51.2 or G50.2 was commanded when the tool axis was set to the linear axis (the parameter "#1017 rot" = 0).

Remedy

- Check and correct the G code address in the program.
- Check the parameter setting values.

P34	Illegal G code	1

Details

An attempt has been made to change the current system mode (G20, G21) with another G20 or G21 command.

Remedy

- Check the program.

P35 Setting value range over

Details

The setting range for the addresses has been exceeded.

Remedy

- Correct the program.

P36 Program end error

Details

"EOR" has been read during tape and memory mode.

Remedy

- Enter the M02 and M30 command at the end of the program.
- Enter the M99 command at the end of the subprogram.

P37 O, N number zero

Details

"0" has been specified for program or sequence No.

Remedy

- Designate program Nos. within a range from 1 to 99999999.
- Designate sequence Nos. within a range from 1 to 99999999.

P38 No spec: Add. Op block skip

Details

"/n" has been issued while the optional block skip addition is not in the specifications.

Remedy

- Check the specifications.

P39 No specifications

Details

- A non-specified G code was commanded.
- The selected operation mode is out of the option setting.
- The selected operation mode is out of the parameter setting.

Remedy

- Check the specifications.
- Check the parameter setting values.

P40 Pre-read block error

Details

When tool radius compensation is executed, there is an error in the pre-read block and so the interference check is disabled.

Remedy

- Reconsider the program.

P45 G code combination error

Details

The combination of G codes in a block is inappropriate.
A part of unmodal G codes and modal G codes cannot be commanded in a same block.

Remedy

Correct the combination of G codes.
Separate the incompatible G codes into different blocks.

P46 Cutting command invalid

Details

A travel command other than G00 (positioning) was issued when all the following conditions were met.

- Parameter "#1247 set19/bit0" (Movement by tool length compensation command) is "1"
- Multiple-axis synchronization control is active
- Tool length compensation is not yet applied after G43 H** command (without axis position command)

Remedy

- When using tool length compensation during Multiple-axis synchronization control, issue a positioning command (G00) before performing cutting.

P48 Restart pos return incomplete

Details

A travel command was issued before the execution of the block that had been restart-searched.

Remedy

- Carry out program restart again.
  Travel command cannot be executed before the execution of the block that has been restart-searched.

P49 Invalid restart search

Details

Restart search was attempted for a block that comes after any of the following command blocks:

3-dimensional circular interpolation, cylindrical interpolation, polar coordinate interpolation, milling interpolation, tool center point control, tool cutting point control, simple tool center point control, inclined surface machining, simple inclined surface machining, workpiece installation error compensation, axis name switch, mixed control, sub part system control I, sub part system control II, direct command mode, arbitrary axis exchange control, or orbit machining.

Remedy

- Modify the position for restart search.

P50 No spec: Inch/Metric change

Details

Inch/Metric changeover (G20/G21) command was issued while the function is out of specifications.

Remedy

- Check the specifications.

P60 Compensation length over

Details

The commanded movement distance is excessive (over 2^{31}).

Remedy

- Correct the command range for the axis address.

P61 No spec: Unidirectional posit.

Details

Unidirectional positioning (G60) was commanded while the function is out of specifications.

Remedy

- Check the specifications.

P62 No F command

Details

- No feed rate command has been issued.
- There is no F command in the cylindrical interpolation or polar coordinate interpolation immediately after the G95 mode is commanded.

Remedy

- The default movement modal command at power ON is G01. This causes the machine to move without a G01 command if a movement command is issued in the program, and an alarm results. Use an F command to specify the feed rate.
- Specify F with a thread lead command.

P65 No spec: High speed mode 3

Details

High-speed mode III (G05 P3) was commanded while the function is out of specifications.

Remedy

- Check whether the specifications are provided for the high-speed mode III.

P67 F value is exceeding the limit

Details

F's value in an F or ",F" command is exceeding the command range.

Remedy

- Check and correct F or ",F" command in the program.

P70 Arc end point deviation large

Details

- There is an error in the arc start and end points as well as in the arc center.
- The difference of the involute curve through the start point and the end point is large.
- When arc was commanded, one of the two axes configuring the arc plane was a scaling valid axis.

Remedy

- Correct the numerical values of the addresses that specify the start and end points, arc center as well as the radius in the program.
- Correct the "+" and "-" directions of the address numerical values.
- Check for the scaling valid axis.

P71 Arc center error

Details

- An arc center cannot be obtained in R-specified circular interpolation.
- A curvature center of the involute curve cannot be obtained.

Remedy

- Correct the numerical values of the addresses in the program.
- Correct the start and end points if they are inside of the base circle for involute interpolation. When carrying out tool radius compensation, make sure that the start and end points after compensation will not be inside of the base circle for involute interpolation.
- Correct the start and end points if they are at an even distance from the center of the base circle for involute interpolation.

P72 No spec: Herical cutting

Details

A helical command has been issued though it is out of specifications.

Remedy

- Check whether the specifications are provided for the helical cutting.
- An Axis 3 command has been issued by the circular interpolation command. If there is no helical specification, move the linear axis to the next block.

P73 No spec: Spiral cutting

Details

A spiral command was issued though it is out of specifications.

Remedy

- Issue the G02.1 and G03.1 commands for circular interpolation.
- Check whether the specifications are provided for the spiral cutting.

P74 Can't calculate 3DIM arc

Details

The 3-dimension circular cannot be obtained because the end block was not specified during 3-dimension circular interpolation supplementary modal.
The 3-dimension circular cannot be obtained due to an interruption during 3-dimension circular interpolation supplementary modal.

Remedy

- Correct the program.

P75 3DIM arc illegal

Details

An illegal G code was issued during 3-dimension circular interpolation modal.
Otherwise, 3-dimension circular interpolation command was issued during a modal for which a 3-dimension circular interpolation command cannot be issued.

Remedy

- Correct the program.

P76 No spec: 3DIM arc interpolat

Details

G02.4/G03.4 was commanded though there is no 3-dimension circular interpolation specification.

Remedy

- Check the specifications.

P90 No spec: Thread cutting

Details

A thread cutting command was issued though it is out of specifications.

Remedy

- Check the specifications.

P93 Illegal pitch vaule

Details

An illegal thread lead (thread pitch) was specified at the thread cutting command.

Remedy

- Correct the thread lead for the thread cutting command.

P100 No spec: Cylindric interpolat

Details

A cylindrical interpolation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P110 Plane select during figure rot

Details

Plane selection (G17/G18/G19) was commanded during figure rotation.

Remedy

- Correct the machining program.

P111 Plane selected while coord rot

Details

Plane selection commands (G17, G18, G19) were issued during a coordinate rotation was being commanded.

Remedy

- Always command coordinate rotation cancel after the coordinate rotation command, and then issue a plane selection command.

P112 Plane selected while R compen

Details

- Plane selection commands (G17, G18, G19) were issued while tool radius compensation (G41, G42) and nose R compensation (G41, G42, G46) commands were being issued.
- Plane selection commands were issued after completing nose R compensation commands when there were no further axis movement commands after G40, and compensation has not been cancelled.

Remedy

- Issue plane selection commands after completing (axis movement commands issued after G40 cancel command) tool radius compensation and nose R compensation commands.

P113 Illegal plane select

Details

The circular command axis does not correspond to the selected plane.

Remedy

- Select a correct plane before issuing a circular command.

P114 Plane axis command error

Details

- The axis specified in the fixed cycle for turning machining does not coincide with the selected plane.
- One or both of the selected plane axes have no travel when the fixed cycle for turning machining is commanded.

Remedy

- Correct the plane selection before issuing the fixed cycle for turning machining.
- Correct the program so that the two axes of the selected plane are involved in the movement of the fixed cycle for turning machining.

P120 No spec: Feed per rotation

Details

Feed per rotation (G95) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P121 F0 command during arc modal

Details

F0 (F 1-digit feed) was commanded during the arc modal (G02/G03).

Remedy

- Correct the machining program.

P122 No spec: Auto corner override

Details

An auto corner override command (G62) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the G62 command from the program.

P123 No spec: High-accuracy control

Details

High-accuracy control command was issued though it is out of specifications.

Remedy

- Check the specifications.

P124 No spec: Inverse time feed

Details

- The inverse time option is not provided.

Remedy

- Check the specifications.

P125 G93 mode error

Details

- The issued G code command is illegal during G93 mode.
- G93 command was issued during a modal for which inverse time feed cannot be performed.

Remedy

- Correct the program.

P126 Invalid cmnd in high-accuracy

Details

An illegal command was issued during the high-accuracy control mode.

- A G code group 13 command was issued during the high-accuracy control mode.
- Milling, cylindrical interpolation or polar coordinate interpolation was commanded during the high-accuracy control mode.

Remedy

- Correct the program.

P127 No spec: SSS Control

Details

The SSS control valid parameter has been set although there is no SSS control specification.

Remedy

- Check the specifications. Set "#8090 SSS ON" to "0" when there is no SSS control specification.

P128 Machin condtn select I disable

Details

Machining condition selection I was commanded during the mode where the selection command is unavailable.

Remedy

- Check the program and see whether any unavailable mode is included during machining condition selection I. If any, cancel that mode.

P129 Hi-speed Hi-accuracy both ON

Details

Both the high-accuracy control mode and high-speed machining mode are commanded simultaneously in a part system where the simultaneous use of the two modes is disabled.

Remedy

- Correct the setting of "#8040 High-speed high-accuracy control-enabled part system".
- Correct the machining program so that the high-accuracy control mode is not used together with high-speed machining mode.

P130 2nd M function code illegal

Details

The 2nd miscellaneous function address, commanded in the program, differs from the address set in the parameters.

Remedy

- Correct the 2nd miscellaneous function address in the program.

P131 No spec: Cnst surface ctrl G96

Details

A constant surface speed control command (G96) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Issue a rotation speed command (G97) instead of the constant surface speed control command (G96).

P132 Spindle rotation speed S=0

Details

No spindle rotation speed command has been issued.

Remedy

- Correct the program.

P133 Illegal P-No. G96

Details

The illegal No. was specified for the constant surface speed control axis.

Remedy

- Correct the parameter settings and program that specify the constant surface speed control axis.

P134 G96 Clamp Err.

Details

The constant surface speed control command (G96) was issued without commanding the spindle speed clamp (G92/G50).

Remedy

Press the reset key and carry out the remedy below.

- Check the program.
- Issue the G92/G50 command before the G96 command.
- Command the constant surface speed cancel (G97) to switch to the rotation speed command.

P135 Spindle select command illegal

Details

This program error is generated in Multiple-spindle Control I. This alarm is generated when spindle selection by address P is valid ("#1300 ext36/bit4" is "1") and either when:

- There is no "P" address in combination with address S, M3, M4 or M5, while parameter "#1297 ext33/bit0" is "0";
- A spindle not defined by parameter "#3199 spCode" is specified;
- An address not allowed in the same block with a "P" address is programmed;
- The same spindle code has been set for some spindles with parameter "#3199 spCode".

Remedy

- Check the program.
- Check the parameter.

P140 No spec: Pos compen cmd

Details

The position compensation command (G45 to G48) is out of specifications.

Remedy

- Check the specifications.

P141 Pos compen during rotation

Details

Position compensation was commanded during the figure rotation or coordinate rotation command.

Remedy

- Correct the program.

P142 Pos compen invalid arc

Details

Position compensation cannot be executed with the issued arc command.

Remedy

- Correct the program.

P150 No spec: Nose R compensation

Details

- Tool radius compensation commands (G41 and G42) were issued though they are out of specifications.
- Nose R compensation commands (G41, G42, and G46) were issued though they are out of specifications.

Remedy

- Check the specifications.

P151 Radius compen during arc mode

Details

A compensation command (G40, G41, G42, G43, G44, or G46) has been issued in the arc modal (G02 or G03).

Remedy

- Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block.
  (Set the modal to linear interpolation.)

P152 No intersection

Details

- In interference block processing during execution of a tool radius compensation (G41 or G42) or nose R compensation (G41, G42, or G46) command, the intersection point after one block is skipped cannot be determined.
- The 3-dimensional tool radius compensation value (tool vertical direction compensation) (G41.2 or G42.2) cannot be determined.

Remedy

- Correct the program.

P153 Compensation interference

Details

An interference error has occurred while the tool radius compensation command (G41 or G42) or nose R compensation command (G41, G42 or G46) was being executed.

Remedy

- Correct the program.

P154 No spec: 3D compensation

Details

A three-dimensional compensation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P155 Fixed cyc exec during compen

Details

A fixed cycle command has been issued in the radius compensation mode.

Remedy

- Issue a radius compensation cancel command (G40) to cancel the radius compensation mode that has been applied since the fixed cycle command was issued.

P156 R compen direction not defined

Details

A shift vector with undefined compensation direction was found at the start of G46 nose R compensation.

Remedy

- Change the vector to that which has the defined compensation direction.
- Change the tool to that which has a different tip point No.

P157 R compen direction changed

Details

During G46 nose R compensation, the compensation direction is reversed.

Remedy

- Change the G command to that which allows the reversed compensation direction (G00, G28, G30, G33, or G53).
- Change the tool to that which has a different tip point No.
- Enable "#8106 G46 NO REV-ERR".

P158 Illegal tip point

Details

An illegal tip point No. (other than 1 to 8) was found during G46 nose R compensation.

Remedy

- Correct the tip point No.

P161 No spec: 3D tool R comp

Details

3-dimensional tool radius compensation (tool vertical direction compensation) is not included in the specifications.

Remedy

- Check the specifications.

P162 Disable Cmd in 3D tool R comp

Details

A command (G or T command, etc.) was issued during 3-dimensional tool radius compensation (tool vertical direction compensation), although it is disabled during the compensation.

Remedy

- Cancel 3-dimensional tool radius compensation (tool vertical direction compensation).

P163 3D tool R comp is disabled

Details

3-dimensional tool radius compensation (tool vertical direction compensation) was commanded in a mode where the command is disabled.

Remedy

- Cancel the mode that disables the command.

P169 Tool offset amount zero

Details

The offset amount designated with the compensation No. (D** or H**) in a compensation command (*1) is set to zero.

(*1) Compensation command means the followings:

- Tool radius compensation (G41, G42, G41.2, G42.2, G43.8, G43.9), the address D
- Tool length compensation (G43, G44, G43.1, G43.4, G43.5, G43.7, G43.8, G43.9), the address H
- Tool position compensation (G45, G46, G47, G48), the address D

Remedy

- Check the offset amount of the compensation No.
- Correct the program.

P170 No offset number

Details

- No compensation No. (D**, T**, H**) command was given when the radius compensation (G41, G42, G43 or G46) command was issued. Otherwise, the compensation No. is larger than the number of sets in the specifications.
- H99 or D99 is commanded with the parameter "#1227 aux11/bit1" enabled when the length compensation method and radius compensation method are set to "1" or "2" for the M system tool life management II.

Remedy

- Add the compensation No. command to the compensation command block.
- Check the number of sets for the tool compensation Nos. and correct the compensation No. command to be within the number of sets.
- H99 and D99 commands cannot be used when the length compensation method and radius compensation are set to "1" or "2". Set the length compensation method and radius compensation method to "0".

P171 No spec:Comp input by prog G10

Details

Compensation data input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P172 G10 L number error

Details

An address of G10 command is not correct.

Remedy

- Correct the address L No. of the G10 command.

P173 G10 P number error

Details

The compensation No. at the G10 command is not within the permitted number of sets in the specifications.

Remedy

- Check the number of sets for the tool compensation Nos. and correct the address P designation to be within the number of sets.

P174 No spec:Comp input by prog G11

Details

Compensation data input by program cancel (G11) was commanded though there is no specification of compensation data input by program.

Remedy

- Check the specifications.

P177 Tool life count active

Details

Registration of tool life management data with G10 was attempted when the "usage data count valid" signal was ON.

Remedy

- The tool life management data cannot be registered during the usage data count. Turn the "usage data count valid" signal OFF.

P178 Tool life data entry over

Details

The number of registration groups, total number of registered tools or the number of registrations per group exceeded the range in the specifications.

Remedy

- Correct the number of registrations.

P179 Illegal group No.

Details

- A duplicate group No. was found at the registration of the tool life management data with G10.
- A group No. that was not registered was designated during the T****99 command.
- An M code command, which must be issued as a single command, coexists in the same block as that of another M code command.
- The M code commands set in the same group exist in the same block.

Remedy

- Register the estimated tool life data/cumulative usage data once for one group: commanding with a duplicate group No. is not allowed.
- Correct to the group No.

P180 No spec: Drilling cycle

Details

A fixed cycle command (G72 - G89) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Correct the program.

P181 No spindle command (Tap cycle)

Details

Spindle rotation speed (S) has not been commanded in synchronous tapping.

Remedy

- Command the spindle rotation speed (S) in synchronous tapping.
- When "#8125 Check Scode in G84" is set to "1", enter the S command in the same block where the synchronous tapping command is issued.

P182 Synchronous tap error

Details

- Connection to the main spindle unit was not established.
- The synchronous tapping was attempted with the spindle not serially connected under the multiple-spindle control I.
- Synchronous tapping command was given to the analog spindle while analog spindle synchronous tapping was unavailable.
- Punch Tap cycle command was given with no-load operation mode during manual arbitrary reverse run mode.

Remedy

- Check connection to the main spindle.
- Check that the main spindle encoder exists.
- Set 1 to the parameter #3024 (sout).
- Correct the program.
- Enable actual cutting mode to command Punch Tap cycle with manual arbitrary reverse run mode.

P183 No pitch/thread number

Details

The pitch or number of threads has not been commanded in the tap cycle of a fixed cycle for drilling command.

Remedy

- Specify the pitch data and the number of threads by F or E command.

P184 Pitch/thread number error

Details

- The pitch or the number of threads per inch is illegal in the tap cycle of the fixed cycle for drilling command.
- The pitch is too small for the spindle rotation speed.
- The thread number is too large for the spindle rotation speed.

Remedy

- Correct the pitch or the number of threads per inch.

P185 No spec: Sync tapping cycle

Details

Synchronous tapping cycle (G84/G74) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P186 Illegal S cmnd in synchro tap

Details

S command was issued during synchronous tapping modal.

Remedy

- Cancel the synchronous tapping before issuing the S command.

P188 Hole edge chamfering invalid

Details

A G185 command was issued when one of the following conditions was true, which disables a hole edge chamfering command.

- The selected plane is not G19 plane.
- The configuration of parameters "#1026 base_ I" through "#1031 aux_K" is one that does not allow hole edge chamfering cycle.
- The "Longitudinal hole drilling axis selection" signal (YCD4) is ON.
- The axis name switch (G111) modal is active.

Remedy

- Ensure that G19 plane is selected when G185 is issued.
- Turn off the "Longitudinal hole drilling axis selection" signal (YCD4).
- Cancel axis name switch (G111) before issuing G185.

P190 No spec: Turning cycle

Details

A lathe cutting cycle command was issued though it is out of specifications.

Remedy

- Check the specification.
- Delete the lathe cutting cycle command.

P191 Taper length error

Details

In the lathe cutting cycle, the specified length of taper section is illegal.

Remedy

- Set the smaller radius value than the axis travel amount in the lathe cycle command.

P192 Chamfering error

Details

Chamfering in the thread cutting cycle is illegal.

Remedy

- Set a chamfering amount not exceeding the cycle.

P199 Tool selection is incorrect

Details

A turning tool shape compensation was commanded while a tool other than turning tool has been selected.

Remedy

- Select "Lathing" for tool type in tool management data before giving the turning tool shape compensation command.

P200 No spec: MRC cycle

Details

The compound type fixed cycle for turning machining I (G70 to G73) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P201 Program error (MRC)

Details

- The subprogram, called with a compound type fixed cycle for turning machining I command, has at least one of the following commands: reference position return command (G27, G28, G29, G30); thread cutting (G33, G34); fixed cycle skip-function (G31, G31.n).
- An arc command was found in the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

Remedy

- Delete G27, G28, G29, G30, G31, G33, G34, and fixed cycle G codes from the subprogram called with the compound type fixed cycle for turning machining I commands (G70 to G73).
- Delete G02 and G03 from the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

P202 Block over (MRC)

Details

The number of blocks in the shape program of the compound type fixed cycle for turning machining I is over 50 or 200 (the maximum number differs according to the model).

Remedy

- Set a 50/200 or less value for the number of blocks in the shape program called by the compound type fixed cycle for turning machining I commands (G70 to G73). (The maximum number differs according to the model).

P203 D cmnd figure error (MRC)

Details

A proper shape will not obtained by executing the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

Remedy

♦Correct the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

P204 MRC command error

Details

A command value of the compound type fixed cycle for turning machining (G70 to G76) is illegal.

Remedy

The alarm has the following possible causes. Correct the compound type fixed cycle for turning machining commands (G70 to G76).

♦G70 - G73

(1) P or Q is omitted.

♦G71

(1) The cut amount per cycle (programmed or specified by "#8051 G71 THICK") is greater than the cut depth for the finished shape.

(2) The cut amount per cycle (programmed or specified by "#8051 G71 THICK") is smaller than double the parameter "#8017 G71 DELTA-D".

♦G74/G75

(1) X/Z is programmed, but the tool shift amount is "0" or unspecified.

(2) The tool shift amount is greater than the travel amount of X/Z.

(3) The escape amount is greater than the tool shift amount.

(4) The return amount is greater than the cut depth.

(5) The cut depth is greater than the hole depth.

♦G76

(1) The tool nose angle (thread angle) is outside of the specified range.

(2) X or Z is omitted.

(3) For X or Z, the coordinates of the start point are the same as of the end point.

(4) The thread is larger than the movement of X axis to the thread bottom.

P210 No spec: Pattern cycle

Details

A compound type fixed cycle for turning machining II (G74 to G76) command was commanded though it is out of specifications.

Remedy

♦Check the specifications.

P220 No spec: Special fixed cycle

Details

There are no special fixed cycle specifications.

Remedy

♦Check the specifications.

P221 No. of special fixed holes = 0

Details

"0" has been specified for the number of holes in special fixed cycle mode.

Remedy

♦Correct the program.

P222 G36 angle error

Details

A G36 command specifies "0" for angle intervals.

Remedy

- Correct the program.

P223 G12/G13 radius error

Details

The radius value specified with a G12 or G13 command is below the compensation amount.

Remedy

- Correct the program.

P224 No spec: Circular (G12/G13)

Details

There are no circular cutting specifications.

Remedy

- Check the specifications.

P225 Illegal character input

Details

- Illegal character string was commanded during character string input.
- Word count exceeded the command range during character string input.

Remedy

- Correct character string.

  Sub error number indicates which character is causing the alarm.

  If the word count exceeds the command range, sub error number will be "0".

P229 Illegal arc rad. of engraving

Details

The diameter of arc for the text is smaller than half the character height or than the character spacing.

Remedy

- Correct the values programmed with the addresses H, I, J and L.

P230 Subprogram nesting over

Details

Over 10 times of subprogram calls have been done in succession from a subprogram.

- A M198 command was found in the program in the data server.

Remedy

- Correct the program so that the number of subprogram calls does not exceed 10 times.

P231 No sequence No.

Details

The sequence No., commanded at the return from the subprogram or by GOTO in the subprogram call, was not set.

Remedy

- Specify the sequence Nos. in the call block of the subprogram.

P232 No program No.

Details

- The machining program has not been found when the machining program is called.

Remedy

- Enter the machining program.
- Check the subprogram storage destination parameters.
- Ensure that the external device (including SD card/USB memory) that contains the file is mounted.

P235 Program editing

Details

Operation was attempted for the file under program editing.

Remedy

- Execute the program again after completion of program editing.

P240 No spec: Variable commands

Details

A variable command (with #) was issued though it is out of specifications.

Remedy

- Check the specifications.

P241 No variable No.

Details

The variable No. commanded is out of the range specified in the specifications.

Remedy

- Check the specifications.
- Correct the program variable No.

P242 = not defined at vrble set

Details

The "=" sign has not been commanded when a variable is defined.

Remedy

- Designate the "=" sign in the variable definition of the program.

P243 Can't use variables

Details

- An invalid variable has been specified in the left or right side of an operation expression.
- Assignment to a write-protected variable has been commanded.

Remedy

- Correct the program.

P244 Invalid set date or time

Details

Date or time was set earlier than current date or time in the system variables (#3011, #3012) when the system lock was valid.

Remedy

- Date or time cannot be changed.
- Correct the program.

P245 Tool No. error

Details

- Tool data read/write command has been executed without selecting the tool command method.
- Tool command method (#68000) or tool selection No. (#68001) is incorrect.
  (1) The tool specified as "Tool in use" is not installed.
  (2) Any unregistered tool No. has been designated.
  (3) Tool selection No. (#68001) has not been designated.
- Write of "Tool No." has been commanded using #68001 while tool No. is being designated.
- Write of "Tool No." has been commanded for an already registered tool No.

Remedy

- Check the program to make sure that the tool command method (#68000) and tool selection No. (#68001) are correct.
- Make sure, if you wish to designate a tool in use, that the said tool No. is nonzero and is already registered on the tool management screen.
- Make sure, if you wish to designate a tool No., that the tool No. registered on the tool management screen has been commanded using #68001.
- #68001 is unable to write the same tool No. as that already registered on the tool management screen.

P246 Ext common var file not found

Details

The extended common variable data file for the specified number does not exist.

Remedy

- Correct the extended common variable data file number in the machining program.
- Ensure that an SD card is inserted.

P247 Ext common var file not loaded

Details

Extended common variable data file is not loaded.

Remedy

- Load the extended common variable data file.
- Ensure that an SD card is inserted.

P248 Ext com vrble file load invld

Details

Extended common variable data file cannot be loaded, because the number of real/integer data in the file is different from the number of real/integer data set in the parameter.

Remedy

- Load the extended common variable data file with the same allocation as the number of real/integer data set in the parameter.

P249 Position data unreadable

Details

An attempt was made to pre-read a system variable (position data) that cannot be read during travel without waiting for the end point of the previous block to be reached.

Remedy

- Use position data that can be pre-read.
- Program the reading of the system variable (position data) and a travel command in separate blocks.

P250 No spec: Figure rotation

Details

Figure rotation (M98 I_J_P_H_L_) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P251 Figure rotation overlapped

Details

Figure rotation command was issued during figure rotation.

Remedy

- Correct the program.

P252 Coord rotate in fig. rotation

Details

A coordinate rotation related command (G68, G69) was issued during figure rotation.

Remedy

- Correct the program.

P260 No spec: Coordinates rotation

Details

A coordinate rotation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P261 G code illegal (Coord rot)

Details

Another G code or a T command has been issued in the block of coordinate rotation command.

Remedy

- Correct the program.

P262 Illegal modal (Coord rot)

Details

A coordinate rotation command has been issued during modal in which coordinate rotation is not allowed.

Remedy

- Correct the program.

P270 No spec: User macro

Details

A macro specification was commanded though it is out of specifications.

Remedy

- Check the specifications.

P271 No spec: Macro interrupt

Details

A macro interruption command has been issued though it is out of specifications.

Remedy

- Check the specifications.

P272 NC and macro texts in a block

Details

An executable statement and a macro statement exist together in the same block.

Remedy

- Place the executable statement and macro statement in separate blocks in the program.

P273 Macro call nesting over

Details

The number of macro call nests exceeded the limit imposed by the specifications.

Remedy

- Correct the program so that the macro calls do not exceed the limit imposed by the specifications.

P275 Macro argument over

Details

The number of argument sets in the macro call argument type II has exceeded the limit.

Remedy

- Correct the program.

P276 Illegal G67 command

Details

A G67 command was issued though it was not during the G66 command modal.

Remedy

- Correct the program.
- Issue G66 command before G67 command, which is a call cancel command.

P277 Macro alarm message

Details

An alarm command has been issued in #3000.

Remedy

- Refer to the operator messages on the diagnosis screen.
- Refer to the instruction manual issued by the machine tool builder.

P280 Brackets [ ] nesting over

Details

Over five times have the parentheses "[" or "]" been used in a single block.

Remedy

- Correct the program so that the number of "[" or "]" is five or less.

P281 Brackets [ ] not paired

Details

A single block does not have the same number of commanded parentheses "[" as that of "]".

Remedy

- Correct the program so that "[" and "]" parentheses are paired up properly.

P282 Calculation impossible

Details

The arithmetic formula is incorrect.

Remedy

- Correct the formula in the program.

P283 Divided by zero

Details

The denominator of the division is zero.

Remedy

- Correct the program so that the denominator for division in the formula is not zero.

P288 IF EXCESS

Details

The multiplicity of the IF statement exceeded 10 times.

Remedy

- The program is reviewed so that the multiplicity of the IF statement does not exceed 10 times.

P289 IF-ENDIF MMC.

Details

IF and ENDIF are not in pairs.
THEN/ELSE is ordered in the absence of IF command.

Remedy

- The program is reviewed so that IF and ENDIF become pairs.
- Before THEN/ELSE command, IF[<conditional expression>]command is given.

P290 IF sentence error

Details

There is an error in the "IF[<conditional>]GOTO(" statement.

Remedy

- Correct the program.

P291 WHILE sentence error

Details

There is an error in the "WHILE[<conditional>]DO(-END(" statement.

Remedy

- Correct the program.

P292 SETVN sentence error

Details

There is an error in the "SETVN(" statement when the variable name setting was made.

Remedy

- Correct the program.
- The number of characters in the variable name of the SETVN statement must be 7 or less.

P293 DO-END nesting over

Details

The number of DO-END nesting levels in the "WHILE[<conditional>]DO(-END(" statement has exceeded 27.

Remedy

- Correct the program so that the nesting levels of the DO-END statement does not exceed 27.

P294 DO and END not paired

Details

The DOs and ENDs are not paired off properly.

Remedy

- Correct the program so that the DOs and ENDs are paired off properly.

P295 WHILE/GOTO in tape

Details

There is a WHILE or GOTO statement on the tape during tape operation.

Remedy

- Apply memory mode operation instead of tape mode that does not allow the execution of the program with a WHILE or GOTO statement.

P296 No address (macro)

Details

A required address has not been specified in the user macro.

Remedy

- Correct the program.

P297 Address-A error

Details

The user macro does not use address A as a variable.

Remedy

- Correct the program.

P298 G200-G202 cmnd in tape

Details

User macro G200, G201, or G202 was specified during tape or MDI mode.

Remedy

- Correct the program.

P300 Variable name illegal

Details

The variable names have not been commanded properly.

Remedy

- Correct the variable names in the program.

P301 Variable name duplicated

Details

A duplicate variable name was found.

Remedy

- Correct the program so that no duplicate name exists.

P310 Not use GMSTB macro code

Details

G, M, S, T, or B macro code was called during fixed cycle.

Remedy

- Correct the program.
- Correct the parameter settings.

P350 No spec: Scaling command

Details

The scaling command (G50, G51) was issued though it is out of specifications.

Remedy

- Check the specifications.

P360 No spec: Program mirror

Details

A mirror image command (G50.1 or G51.1) has been issued though the programmable mirror image specifications are not provided.

Remedy

- Check the specifications.

P370 No spec: Facing t-post MR

Details

The facing turret mirror image specifications are not provided.

Remedy

- Check the specifications.

P371 Facing t-post MR illegal

Details

- Mirror image for facing tool posts was commanded to an axis in external mirror image or parameter mirror image.
- The commanded mirror image for facing tool posts enables the mirror image for a rotary axis.

Remedy

- Correct the program.
- Correct the parameter settings.

P380 No spec: Corner R/C

Details

The corner R/C was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the corner chamfering/corner rounding command in the program.

P381 No spec: Arc R/C

Details

Corner chamfering II or corner rounding II was commanded in the arc interpolation block though it is out of specifications.

Remedy

- Check the specifications.

P382 No corner movement

Details

The block next to corner chamfering/ corner rounding is not a travel command.

Remedy

- Replace the block succeeding the corner chamfering/ corner rounding command by G01 command.

P383 Corner movement short

Details

The travel distance in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance.

P384 Corner next movement short

Details

The travel distance in the following block in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance in the following block.

P385 Corner during G00/G33

Details

A block with corner chamfering/corner rounding was given during G00 or G33 modal.

Remedy

- Correct the program.

P390 No spec: Geometric

Details

A geometric command was issued though it is out of specifications.

Remedy

- Check the specifications.

P391 No spec: Geometric arc

Details

There are no geometric IB specifications.

Remedy

- Check the specifications.

P392 Angle < 1 degree (GEOMT)

Details

The angular difference between the geometric line and line is 1° or less.

Remedy

- Correct the geometric angle.

P393 Inc value in 2nd block (GEOMT)

Details

The second geometric block has a command with an incremental value.

Remedy

- Issue a command with an absolute value in the second geometric block.

P394 No linear move command (GEOMT)

Details

The second geometric block contains no linear command.

Remedy

- Issue the G01 command.

P395 Illegal address (GEOMT)

Details

The geometric format is invalid.

Remedy

- Correct the program.

P396 Plane selected in GEOMT ctrl

Details

A plane switching command was issued during geometric command processing.

Remedy

- Complete the plane switching command before geometric command processing.

P397 Arc error (GEOMT)

Details

In geometric IB, the arc end point does not contact or intersect the start point of the next block.

Remedy

- Correct the first and second commands containing a circular arc command for geometric.

P398 No spec: Geometric1B

Details

A geometric command was issued though the geometric IB specifications are not provided.

Remedy

- Check the specifications.

P411 Illegal modal G111

Details

- G111 was issued during milling mode.
- G111 was issued during nose R compensation mode.
- G111 was issued during constant surface speed.
- G111 was issued during mixed control (cross axis control).
- G111 was issued during fixed cycle.
- G111 was issued during polar coordinate interpolation.
- G111 was issued during cylindrical interpolation mode.

Remedy

- Before commanding G111, cancel the following commands.
- Milling mode
- Nose R compensation
- Constant surface speed
- Mixed control (cross axis control)
- Fixed cycle
- Polar coordinate interpolation
- Cylindrical interpolation

P412 No spec: Axis name switch

Details

Axis name switch (G111) was issued though it is out of specifications.

Remedy

- Check the specifications.

P420 No spec: Para input by program

Details

Parameter input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P421 Parameter input error

Details

- The specified parameter No. or set data is illegal.
- An illegal G command address was input in parameter input mode.
- A parameter input command was issued during fixed cycle modal or nose R compensation.
- G10 L50, G10 L70, G10 L100, G11 were not commanded in independent blocks.

Remedy

- Correct the program.

P422 Tool/Work shape input error

Details

- G10 L100, G10 L101 or G11 has been given together with any other command in a block.
- Address P or T has been omitted from G10 L100.
- Address C has been omitted from G10 L101.

Remedy

- Correct the program.

P423 R-Navi input error

Details

- G10 L110, G10 L111 or G11 has been given together with any other command in a block.
- Address Q has been omitted from G10 L110.
- Address P, Q or D has been omitted from G10 L111.
- Machining surface parameter input command has been issued for an undefined workpiece.
- An index angle command has been given to any axis other than Z when using the coordinate axis direction setting method.
- Neither the workpiece registration No. nor surface registration No. has been selected when setting a machining surface.

Remedy

- Correct the program.
- Correct the setting of workpiece to machine.

P430 R-pnt return incomplete

Details

- A command was issued to move an axis, which has not returned to the reference position, away from that reference position.
- A command was issued to an axis removal axis.

Remedy

- Execute reference position return manually.
- Disable the axis removal on the axis for which the command was issued.

P431 No spec: 2,3,4th R-point ret

Details

A command for second, third or fourth reference position return was issued though there are no such command specifications.

Remedy

- Check the specifications.

P432 No spec: Start position return

Details

Start position return (G29) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P433 No spec: R-position check

Details

Reference position check (G27) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P434 Compare error

Details

One of the axes did not return to the reference position when the reference position check command (G27) was executed.

Remedy

- Correct the program.

P435 G27 and M commands in a block

Details

An M command was issued simultaneously in the G27 command block.

Remedy

- Place the M code command, which cannot be issued in a G27 command block, in separate block from G27 command block.

P436 G29 and M commands in a block

Details

An M command was issued simultaneously in the G29 command block.

Remedy

- Place the M code command, which cannot be issued in a G29 command block, in separate block from G29 command block.

P438 G52 invalid during G54.1

Details

A local coordinate system command was issued during execution of the G54.1 command.

Remedy

- Correct the program.

P450 No spec: Chuck barrier

Details

The chuck barrier on command (G22) was specified although the chuck barrier is out of specifications.

Remedy

- Check the specifications.

P451 No spec: Stroke chk bef travel

Details

Stroke check before travel (G22/G23) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P452 Limit before travel exists

Details

An illegal command, which places the axis travel start/end point in the prohibited area or moves the axis through the prohibited area, was detected when Stroke check before travel (G22) was commanded.

Remedy

- Correct the coordinate values of the axis address commanded in the program.

P460 Tape I/O error

Details

An error has occurred in the tape reader. Otherwise an error has occurred in the printer during macro printing.

Remedy

- Check the power and cable of the connected devices.
- Correct the I/O device parameters.

P461 File I/O error

Details

- A file of the machining program cannot be read.
- Unable to write to an external file.

Remedy

- In memory mode, the programs stored in memory may have been destroyed. Output all of the programs and tool data and then format the system.
- Ensure that the external device (including SD card/USB memory) that contains the file is mounted.
- Correct the parameter settings for HD operation, SD card operation or USB memory operation.

P462 Computer link commu error

Details

A communication error occurred during the BTR operation.

Remedy

- "L01 Computer link error" is displayed simultaneously. Take the remedy corresponding to the error No.

P470 Invalid cmd in 3D interf check

Details

Any of the following commands has been given during 3D machine interference check.

- Program format switch (G188/G189)
- Diameter/Radius designation switch (G10.9)

Remedy

- Before the command, correct the program or disable interference check.
- Use the PLC I/F method to switch the program format.

P480 No spec: Milling

Details

- Milling was commanded though it is out of specifications.
- Polar coordinate interpolation was commanded though it is out of specifications.

Remedy

- Check the specifications.

P481 Illegal G code (mill)

Details

- An illegal G code was used during the milling mode.
- An illegal G code was used during cylindrical interpolation or polar coordinate interpolation.
- The G07.1 command was issued during the tool radius compensation.

Remedy

- Correct the program.

P482 Illegal axis (mill)

Details

- A rotary axis was commanded during the milling mode.
- Milling was executed though an illegal value was set for the milling axis No.
- Cylindrical interpolation or polar coordinate interpolation was commanded during mirror image.
- Cylindrical interpolation or polar coordinate interpolation was commanded before the tool compensation was completed after the T command.
- G07.1 was commanded when cylindrical interpolation was not possible (there is no rotary axis, or external mirror image is ON).
- An axis other than a cylindrical coordinate system axis was commanded during cylindrical interpolation.

Remedy

- Correct the machining program, parameters and PLC interface signals.

P484 R-pnt ret incomplete (mill)

Details

- Movement was commanded to an axis that had not completed reference position return during the milling mode.
- Movement was commanded to an axis that had not completed reference position return during cylindrical interpolation or polar coordinate interpolation.

Remedy

- Carry out manual reference position return.

P485 Illegal modal (mill)

Details

- The milling mode was turned ON during nose R compensation or constant surface speed control.
- A T command was issued during the milling mode.
- The mode was switched from milling to cutting during tool compensation.
- Cylindrical interpolation or polar coordinate interpolation was commanded during the constant surface speed control mode (G96).
- The command unacceptable in the cylindrical interpolation was issued.
- A T command was issued during the cylindrical interpolation or polar coordinate interpolation mode.
- A movement command was issued when the plane was not selected just before or after the G07.1 command.
- A plane selection command was issued during the polar coordinate interpolation mode.
- Cylindrical interpolation or polar coordinate interpolation was commanded during tool radius compensation.
- The G16 plane in which the radius value of a cylinder is "0" was specified.
- A cylindrical interpolation or polar coordinate interpolation command was issued during coordinate rotation by program.

Remedy

- Correct the program.
- Issue G40 or G97 before issuing G12.1.
- Issue a T command before issuing G12.1.
- Issue G40 before issuing G13.1.
- Specify the radius value of a cylinder other than "0", or specify the X axis's current value other than "0" before issuing G12.1/G16.

P486 Milling error

Details

- Milling interpolation was issued to the rotary axis for which Mirror image by parameter setting or Mirror image by external input is activated.
- Polar coordinate interpolation, cylindrical interpolation or milling interpolation was commanded during mirror image for facing tool posts.
- The start command of the cylindrical interpolation or polar coordinate interpolation was issued during the normal line control.

Remedy

- Correct the program.

P490 Eccentric turn mode cmd invld

Details

The function that cannot be commanded during Eccentric turning mode is commanded.

Remedy

- Correct the machining program.

P491 Eccentric turn cmd invalid

Details

An Eccentric turning command was issued in a mode where an Eccentric turning command cannot be issued.

Remedy

- Correct the machining program.

P492 Eccentric turn shape error

Details

The shape specified by the Eccentric turning command is incorrect.

- The amount of travel in the radial direction of turning is equal to or smaller than (Q - R - ",I") in the Eccentric turning command.
- The command value of address ",I" is equal to or smaller than that of address R for the Eccentric turning command.
- The command value of address ",I" is smaller than that of address ",J" for the Eccentric turning command.

Remedy

- Correct the Eccentric turning command (G2.6/G3.6).

P493 Eccentric turn cmd error

Details

The command does not meet the format for Eccentric turning.

- Address "L", "R" or ",S" is not commanded.
- The command value of address L is either less than "1" or more than "999".
- Address ",S", ",I" or ",J" is not a positive value.
- Address ",I" and ",J" do not have commas.
- The command was issued for base axes other than those that define the selected plane.

Remedy

- Correct the Eccentric turning command (G2.6/G3.6).

P494 Eccentric turning para error

Details

An Eccentric turning command was issued in one of the following conditions:

- The parameters "#1026 base_I" through "#1028 base_K" are not specified or are duplicated.
- The parameter "#8046 Ecc_rotax" is set to "0".
- The parameter "#1017 rot" is not set to "1" or the parameter "#1020 sp_ax" is not set to "1" in the axis with the number specified in parameter "#8046 Ecc_rotax".

Remedy

- Change the parameter.

P501 Cross (G110) impossible

Details

Mixed control command (G110), Arbitrary axis exchange command (G140), Arbitrary axis exchange return command (G141) or Base axis configuration restore command (G142) has been given to a part system that is under any of the following state.

- Cylindrical interpolation mode
- Polar coordinate interpolation mode
- Milling interpolation mode
- Constant surface speed control mode
- Polygon cut mode
- Hob machining mode
- Nose R compensation mode
- Tool radius compensation mode
- During axis name switch
- Fixed cycle for drilling
- Fixed cycle for lathe turning
- Compound-type fixed cycle mode
- Special fixed cycle mode
- Facing turret mirror image mode
- Balance cut mode
- Chuck barrier/Tailstock barrier
- Stroke check before travel
- During Macro modal call (G66.1)
- Hypothetical axis interpolation mode
- Figure rotation mode
- During scaling
- Coordinate rotation by parameter mode
- Polar coordinate command mode
- Normal line control mode
- Circular cutting mode
- Tool direction tool length compensation mode

Remedy

- Correct the program.

P503 Illegal G110 axis

Details

Any of the following commands was given to an axis that is under the following conditions: Mixed control command, Arbitrary axis exchange command, Arbitrary axis exchange return command or Basic axis configuration return command.

- The commanded axis does not exist.
- The command caused the maximum number of axes per part system to be exceeded.
- The command was given to an axis for which axis exchange is disabled.

Remedy

- Correct the program.

P505 Hobbing axis selection illegal

Details

The axis No. set by the hobbing workpiece axis selection signal is not a rotary axis, or it is over the number of NC axes of the part system.

Remedy

- Confirm that the axis set by the hobbing workpiece axis selection signal is a rotary axis and also an NC axis in the part system.

P511 Synchronization M code error

Details

- Two or more synchronization M codes were commanded in the same block.
- The synchronization M code and "!" code were commanded in the same block.
- Synchronization with the M code was commanded in 3rd part system or more. (Synchronization with the M code is valid only in 1st part system or 2nd part system.)

Remedy

- Correct the program.

P520 Control axis superimposition/Designated axis illegal

Details

- The axis designated as the reference or superimposed axis is not available for superimposition.

Remedy

Correct the program.

P521 Illegal synchronization axis

Details

The axis specified as the reference or synchronized axis of synchronization between part systems cannot be synchronized.

Remedy

- Correct the program.

P522 Simple sync ax setting invalid

Details

The main axis for the simple synchronous control has been designated as tap axis for synchronous tapping.

Remedy

- Correct the program.

P530 Decimal point not possible

Details

A decimal point is included in the address for which a decimal point is invalid.

("#1274 ext10/bit0" (Decimal point command valid address))

Remedy

- Remove the decimal point from the address for which a decimal point is invalid.

P540 No spec: G54.2

Details

- G54.2 was commanded when workpiece position offset for rotary axis was OFF.

Remedy

- Check the workpiece position offset for rotary axis.

P544 No spec: Wk instl err cmp

Details

The workpiece installation error compensation function is out of the specifications.

Remedy

- Check the specifications.

P545 Invld cmd in wk instl err cmp

Details

During workpiece installation error compensation, a command impossible to issue (such as G command) was issued.

Remedy

- Check the program. If you wish to issue a command impossible to issue (such as G command) during workpiece installation error compensation, cancel workpiece installation error compensation once.

P546 Wk instl err cmp cmd invalid

Details

- Workpiece installation error compensation was commanded in a G modal in which commanding it is not allowed.
- An illegal G command was issued in the block that has a workpiece installation error compensation command.

Remedy

- Check the program. Also check the G modals which were issued at commanding the workpiece installation error compensation, and cancel illegal ones.
- Issue the G command in a separate block.

P547 Illegal wk instl err cmp cmd

Details

A command in which the rotary axis's travel distance exceeds 180 degrees was issued.

Remedy

- Divide the travel command so that the rotary axis's travel distance per block is less than 180 degrees.

P550 No spec: G06.2(NURBS)

Details

There is no NURBS interpolation option.

Remedy

- Check the specifications.

P551 G06.2 knot error

Details

The knot (k) command value is smaller than the value for the previous block.

Remedy

- Correct the program.
- Specify the knot by monotone increment.

P552 Start point of 1st G06.2 err

Details

The block end point immediately before the G06.2 command and the G06.2 first block command value do not match.

Remedy

- Match the G06.2 first block coordinate command value with the previous block end point.

P554 Invld manual interrupt in G6.2

Details

Manual interruption was executed in a block that applies the G06.2 mode.

Remedy

- Execute the manual interruption in the block that does not apply the G06.2 mode.

P555 Invalid restart during G06.2

Details

Restart was attempted from the block that applies G06.2 mode.

Remedy

- Restart from the block other than in G06.2 mode.

P560 Fairing changeover disabled

Details

- A command to enable the fairing function was given while the smooth fairing function was ON.
- A command to enable the smooth fairing function was given while the fairing function was ON.

Remedy

- Correct the program.

P580 No spec: Axis selection for F

Details

- The option "selection of axis (axes) for feedrate command" is invalid.

Remedy

- Check the specifications.

P581 Axis selection for F invalid

Details

- The command "selection of axis (axes) for feedrate command" has been issued while the mode that disables the command is active.

Remedy

- Correct the program.

P582 Invalid cmd in F ax selection

Details

- The command issued is invalid while selection of axis (axes) for feedrate command is ON.

Remedy

- Correct the program.

P595 Skip axis illegal

Details

- No axis or more than 2 axes was/were issued to torque limitation skip command block.
- An axis during synchronous control was issued as the skip axis.
- An axis during inclined axis control was issued as the skip axis.
- Geometric command, corner R, or corner chamfering was issued in the same block as torque limitation skip command.
- A torque skip was commanded to the axis which is in the constant torque control or the proportional torque stopper control.

Remedy

- Issue only an axis to the skip axis.
- Review the program.

P600 No spec: Auto TLM

Details

An automatic tool length measurement command (G37) was issued though it is out of specifications.

Remedy

- Check the specifications.

P601 No spec: Skip

Details

A skip command (G31) was issued though it is out of specifications.

Remedy

- Check the specifications.

P602 No spec: Multi skip

Details

A multiple skip command (G31.1, G31.2, G31.3 or G31 Pn) was issued though it is out of specifications.

Remedy

- Check the specifications.

P603 Skip speed 0

Details

The skip speed is "0".

Remedy

- Specify the skip speed.

P604 TLM illegal axis

Details

No axis was specified in the automatic tool length measurement block. Otherwise, two or more axes were specified.

Remedy

- Specify only one axis.

P605 T & TLM command in a block

Details

The T code is in the same block as the automatic tool length measurement block.

Remedy

- Specify the T code before the automatic tool length measurement block.

P606 T cmnd not found before TLM

Details

The T code was not yet specified in automatic tool length measurement.

Remedy

- Specify the T code before the automatic tool length measurement block.

P607 TLM illegal signal

Details

The measurement position arrival signal turned ON before the area specified by the D command or "#8006 ZONE d". Otherwise, the signal remained OFF to the end.

Remedy

- Correct the program.

P608 Skip during radius compen

Details

A skip command was issued during radius compensation processing.

Remedy

- Issue a radius compensation cancel (G40) command or remove the skip command.

P610 Illegal parameter

Details

- The parameter setting is not correct.
  - G114.1 was commanded while the spindle synchronization was selected with the PLC interface signal.
  - G110 was commanded while the mixed control (cross axis control) was selected with the PLC interface signal.
  - G125 was commanded while the control axis synchronization between part systems was selected with the PLC interface signal.
  - G126 was commanded while the control axis superimposition was selected with the PLC interface signal.

Remedy

- Correct the settings of "#1514 expLinax (Exponential function interpolation linear axis)" and "#1515 expRotax (Exponential function interpolation rotary axis)".
  - Correct the program.
  - Correct the parameter settings.

P611 No spec: Exponential function

Details

There is no specification for the exponential interpolation.

Remedy

- Check the specifications.

P612 Exponential function error

Details

A travel command for exponential interpolation was issued during mirror image for facing tool posts.

Remedy

- Correct the program.

P650 Sub sys identification # error

Details

- The identification No. specified in the address B of G122 or G144 is that of the part system where G122 is given.
- The identification No. specified in the sub part system control I command (G122) is not set in the parameter "#12049 SBS_no".

Remedy

- Change the address B of G122 or G144 to be any identification No. other than that of the part system where G122 is given.
- Select the identification No. for the sub part system control I command (G122) from among the available Nos.
- Specify the identification No. you wish to use for the sub part system control I command (G122) in the parameter "#12049 SBS_no".

P651 Other G code in sub sys block

Details

G122 or G144 has been given together with any other G code command in a block.

Remedy

- Do not command G122 or G144 together with any other G code command in a block.

P652 Illegal mode (sub part system)

Details

- G144 has been commanded while in either High-Speed Machining Mode or High-Speed High-Accuracy Control.
- G122 or G144 has been commanded while in any of the following modes.
  - User Macro Modal Call (G66/G66.1)
  - Fixed Cycle

Remedy

- Cancel the following modes before commanding G144.
  - High-Speed Machining Mode
  - High-Speed High-Accuracy Control
- Cancel the following modes before commanding G122 or G144.
  - User Macro Modal Call (G66/G66.1)
  - Fixed Cycle

P653 Illegal G code (sub part sys)

Details

High-speed mode command (G5, G5.1) has been given in a sub part system.

Remedy

- Do not use the high-speed mode (G5, G5.1) in a sub part system.

P656 Illegal PLC device

Details

- Specified the device other than R register/D register.
- Specified the odd numbered device when 4 byte is specified.
- Specified the device number that is out of the command range.
- Specified the data length that is out of the command range.
- Specified the bit number that is out of the command range.
- Omitted the device number.
- Omitted the project number after ",P".
- Omitted the data length after ",".
- Omitted the bit number after ".".

Remedy

- Check the program.

P657 PLC Device too much

Details

Multiple assignment expressions which include the PLC direct interface command are commanded to the same block.

Remedy

- Command the assignment command using PLC direct interface by itself.

P700 Illegal command value

Details

Spindle synchronization was commanded to a spindle that is not connected serially.

Remedy

- Correct the program.
- Correct the parameter settings.

P705 Dia/Rad selection cmd invalid

Details

G10.9 was commanded during a modal in which diameter/radius designation selection is not available.

Remedy

- Check the program.

P706 Invld cmd in dia/rad selection

Details

A G code impossible to command was issued during switching between diameter and radius using the diameter/radius designation selection.

Remedy

- Check the program.

P709 No Rapid block overlap

Details

The following commands were executed while there is no option for Rapid traverse block overlap.

- G0.5 command
- Reference position return command while "#1443 G28ol" (Enable G28 rapid traverse block overlap) is enabled
- G0 command while "#12056 I_G0ol" (Enable G00 initial rapid traverse block overlap) is enabled

Remedy

- Check the specification.
- Disable the parameter.

P721 No selected point for return

Details

G26 (Return to selected point) has been given in a program other than a machining interruption program.

Remedy

- Correct the program.

P727 Selected point error

Details

The command order of the selected point is incorrect.

Remedy

- Correct the program.

P728 Invalid cmnd in mach interrupt

Details

The executed command is unavailable in a machining interruption program.

Remedy

- Correct the program.

P729 Invalid cmd before tap retract

Details

The command in the retraction program cannot be executed prior to the tap retraction command (G26).

Remedy

- Place the command after the tap retraction (G26) block.

P730 Invalid end pt for tap retract

Details

The position (end point) given to the drill axis in the tap retraction command (G26) is lower than the R-point (closer to the hole bottom).

Remedy

- Correct the coordinates of the drill axis command.

P740 Invalid cmd in multi-axis sync

Details

The issued command is disabled during multiple-axis synchronization control.

Remedy

- Cancel the multiple-axis synchronization control before issuing the command.

P790 Illegal command in VCC mode

Details

The issued command cannot be used during the vibration cutting mode.

Remedy

- Correct the program.

P791 VCC mode command illegal

Details

Vibration cutting mode start command has been given although the command is disabled during the currently active mode.

Remedy

- Correct the program.

P801 Turning tool offset disabled

Details

Any of the following commands has been given during the G43.7 mode.

- Mirror image by G code
- Mirror image by parameter setting
- Mirror image by external input
- Tool length compensation along the tool axis
- Automatic tool length measurement

A G43.7 command has been given in any of the following modes.

- Mirror image by G code
- Mirror image by parameter setting
- Mirror image by external input
- Fixed cycle for drilling

Remedy

- Check the program. If you use the mirror image, tool length compensation along the tool axis, automatic tool length measurement or fixed cycle for drilling, use a G49 command to cancel the tool position offset.

P802 Nose R compensation disabled

Details

Any of the following commands has been given during nose R compensation for machining center system.

- Mirror image by G code
- Mirror image by parameter setting
- Mirror image by external input

Nose R compensation for machining center system has been executed in any of the following modes.

- Mirror image by G code
- Mirror image by parameter setting
- Mirror image by external input

Remedy

- Check the program. If you use the mirror image, use a G40 command to cancel the nose R compensation for machining center system.

P803 Proceed of prog check disabled

Details

The commanded G code has disabled the program check.

Remedy

Delete the G code that disables the program check, and then retry the program check.
(Note that deleting the G code may affect the operation of the subsequent blocks.)

P811 Invalid cmd in MCR coord TF

Details

A command that is invalid during motion control release (coordinate transformation) was issued.

Remedy

- Correct the program.

P890 Invalid command during F-CUT

Details

An invalid command was executed during F-CUT.

Possible commands are as follows.

- The M code for F-CUT ON/OFF command during beam ON
- The M code for beam ON/OFF command in the same block as the M code for F-CUT ON/OFF command
- A movement G code command other than G00, G01, G02, G03, G28, and G30
- An axis command other than the axes set as base axes I and J
- G04, G64 command

Remedy

- Correct the machining program.

P891 Rotary ax cmd invalid in F-CUT

Details

A rotary axis command was executed while F-CUT was active.

Remedy

- Correct the machining program.

P892 Invalid cmd in laser part sys

Details

A command (G command, etc.) not executable in the part system where the laser processing control was active was issued.

Remedy

- Correct the program.

P893 Simul laser prcsg cond chg cmd

Details

The M code for laser processing condition selection was specified when changing laser processing conditions was not permissible.

- The M code for laser processing condition selection was specified in the same block as the M code for laser beam ON/OFF.

Remedy

- Correct the program.

P900 No spec: Normal line control

Details

A normal line control command (G40.1, G41.1, or G42.1) was issued though it is out of specifications.

Remedy

- Check the specifications.

P901 Normal line control axis G92

Details

A coordinate system preset command (G92) was issued to a normal line control axis during normal line control.

Remedy

- Correct the program.

P902 Normal line control axis error

Details

- The normal line control axis was set to a linear axis.
- The normal line control axis was set to the linear type rotary axis II axis.
- The normal line control axis has not been set.
- The normal line control axis is the same as the plane selection axis.

Remedy

- Correct the normal line control axis setting.

P903 Plane chg in Normal line ctrl

Details

The plane selection command (G17, G18, or G19) was issued during normal line control.

Remedy

- Delete the plane selection command (G17, G18, or G19) from the program of the normal line control.

P910 No spec: tool cutting pnt ctrl

Details

The option for tool cutting point control is not set.

Remedy

- Check the specifications.

P911 Illegal TCPC command

Details

- Tool cutting point control was programmed during a modal where tool cutting point control is not permissible.
- Tool cutting point control was programmed in the same block as a G code that cannot be used with tool cutting point control.

Remedy

- Correct the program.
- Deactivate the function that cannot be used together.

P912 Illegal cmd during TCPC

Details

A G code not executable during tool cutting point control mode was programmed.

Remedy

- Correct the program.
- Deactivate tool cutting point control.

P913 TCPC tool data illegal

Details

Tool cutting point control was programmed with incorrect tool data.

Remedy

- Correct the tool data.
- Modify the parameter.

P920 No spec: 3D coord conv

Details

There is no specification for 3-dimensional coordinate conversion.

Remedy

- Check the specifications.

P921 Illegal G code at 3D coord

Details

The commanded G code cannot be performed during 3-dimensional coordinate conversion modal.

Remedy

- Refer to "Programming Manual" for usable G commands.
- When the parameter "#8158 Init const sur spd" is enabled, disable the parameter or issue the constant surface speed control cancel (G97) command.

P922 Illegal mode at 3D coord

Details

A 3-dimensional coordinate conversion command was issued during a modal for which 3-dimensional coordinate conversion cannot be performed.

Remedy

- Refer to "Programming Manual" for usable G commands.

P923 Illegal addr in 3D coord blk

Details

A G code and G68 was commanded in a block though the G code cannot be commanded with G68.

Remedy

- Refer to "Programming Manual" for usable G commands.

P924 Travel cmd error in 3Dconvert

Details

An incorrect travel command has been given during the 3D coordinate conversion mode.

Remedy

- Correct the program.

P925 End point err in 3D conversion

Details

The end point of a travel command given during "G68.1 ,E1" is not on the tool path.

Remedy

- Correct the program.

P926 3D coordinate cnv format error

Details

There is an error in the 3-dimensional coordinate conversion command.

Check the following:

- Not all IJK commands are described in the 3-dimensional coordinate conversion command block.

Remedy

- Correct the program.

P930 No spec: Tool axis compen

Details

A tool length compensation along the tool axis command was issued though it is out of specifications.

Remedy

- Check the specifications.

P931 Executing tool axis compen

Details

There is a G code that cannot be commanded during tool length compensation along the tool axis.

Remedy

- Correct the program.

P932 Rot axis parameter error

Details

- There is an invalid orthogonal axis name or rotary axis name set in the rotary axis configuration parameters.
- There is an invalid setting related to axis configuration in the rotary axis configuration parameters.
- A function that cannot be commanded when the spindle end offset is set was commanded.

Remedy

- Set the correct value and reboot.
- Check the program.

P934 Invalid axis configuration

Details

- The function cannot be executed under the axis configuration of the part system.
- An unavailable G code was given when "Select specifications of rotation direction parameter" is enabled (#1450/bit3 = 1), and the left-hand screw direction is selected by the rotation direction parameter (#7923, #7933, #7943, #7953 = 1).
- The axis with the axis address H for absolute dimension command is included in the machine configuration.

Remedy

- Correct the program.
- Check and correct the rotary axis configuration parameters so that the function can be executed in the axis configuration.

P940 No spec: Tool tip control

Details

There is no specification for tool tip center control.

Remedy

- Check the specifications.

P941 Invalid T tip control command

Details

A tool tip center control command was issued during a modal for which a tool tip center control command cannot be issued.

Remedy

- Correct the program.

P942 Invalid cmnd during T tip ctrl

Details

A G code that cannot be commanded was issued during tool tip center control.

Remedy

- Correct the program.

P943 Tool posture command illegal

Details

For tool center point control/tool cutting point control type 1, if the signs are different between the start point and end point of the tool-side rotary axis or table base-side rotary axis, the tool base-side rotary axis or the table workpiece-side rotary axis rotation exists in the same block, and does not pass a singular point.
For tool center point control/tool cutting point control type 2, IJK command is incorrect.

Remedy

- Correct the program.

P950 No spec: Tilt face machining

Details

Inclined surface machining option is not supported.

Remedy

- Check the specifications.

P951 Ill cmd in tilt face machining

Details

A forbidden command (G command, etc.) was issued during inclined surface machining.

Remedy

- Check the program. If you want to execute a command (G command, etc.) that is unavailable during inclined surface machining, cancel the inclined surface machining.

P952 Inclined face cut prohibited

Details

Inclined surface machining was commanded during the mode where the machining is unavailable.
Inclined surface machining was commanded during interruption.

Remedy

- Check the program and see whether any unavailable mode is included during inclined surface machining command. If any, cancel that mode.

P953 Tool axis dir cntrl prohibited

Details

Tool axis direction control was commanded during the mode where the control is unavailable.

Remedy

- Check the program and see whether any unavailable mode is included during tool axis direction control. If any, cancel that mode.

P954 Inclined face command error

Details

The address to issue the inclined surface machining command is incorrect.

Remedy

- Check the program.

P955 Inclined face coord illegal

Details

Impossible to define an inclined surface with the values you specified.

Remedy

- Check the program.

P956 G68.2P10 surface not defined

Details

The coordinate system for the machining surface selected with G68.2P10 has not been defined.

Remedy

- Set the machining surface so that the coordinate system can be defined.

P957 Tool axis dir ctrl cmp amt 0

Details

When the tool axis direction control type 2 (G53.6) was commanded, a tool length compensation No. whose compensation amount is 0 was commanded.

Remedy

- Correct the program. Set the tool length compensation amount, or command a tool length compensation No. whose compensation amount is not 0.

P958 Tool axis dir ctrl axis illeg

Details

The command was issued with an angle with which tool axis direction control of 4-axis configuration is not available.

Remedy

- Check the feature coordinate system defined in inclined surface machining command.

P959 No spec: Simple inclined face

Details

Simple inclined surface machining is not included in the specifications.

Remedy

- Check the specifications.

P960 No spec: Direct command mode

Details

G05 P4 was commanded while direct command mode option is OFF.

Remedy

- Check the specifications.

P961 Invalid during dir cmnd mode

Details

- A G code other than G05 P0 was commanded in direct command mode.
- A sequence No. command, F code command, MSTB command or variable command was issued.
- A corner chamfering command or corner R command was issued.
- A travel command was issued to an axis that had not been command in the G05 P4 block.

Remedy

- Check the program.

P962 Dir cmnd mode cmnd invalid

Details

G05 P4 was commanded in a modal where direct command mode is not available.

Remedy

- Check the program.

P963 Illegal direct cmnd mode cmnd

Details

The commanded coordinate value was beyond the maximum travel distance in direct command mode.

Remedy

- Correct the coordinate value in direct command mode.

P990 PREPRO error

Details

Combining commands that required pre-reading (nose R offset, corner chamfering/corner rounding, geometric I, geometric IB, and compound type fixed cycle for turning machining) resulted in eight or more pre-read blocks.

Remedy

- Delete some or all of the combinations of commands that require pre-reading.

P993 Orbit_CMND not allowed

Details

An attempt is made to run a command not allowed during the Orbit machining.

Remedy

- Correct the program.
- Cancel the Orbit machining mode.

P994 Orbit not permitted

Details

An attempt is made to run Orbit machining even though it is not permitted in the current control mode.

Remedy

- Correct the program.
- Cancel the control mode that is not permitted.

13

Smart Safety Observation Alarm (V)

13.1 Smart Safety Observation Error (V01/V02/V03/V04/V05/V06/V07)

V01	**Safety watchdog error**	0001	

Details

Safety function is not carried out in specified cycle.

Remedy

- CPU may be faulty. Contact our service center.

V01	**Cross-check error**	0002	

Details

Each operation result of redundant CPU is different value.

Remedy

- CPU may be faulty. Contact our service center.

V01	**Safe sys internal process err**	0003	Function No.

Details

An error has occurred in the NC's internal process during execution of a safety function.
The screen displays which safety function has been executed at the time of error, using the following numbers.

0001: Safely-limited speed (SLS)
0002: Safely-limited position (SLP)
0003: Safe speed monitor (SSM)
0004: Safe cam (SCA)
0005: Safe operating stop (SOS)
0006: Safe stop 1 (SS1)
0007: Safe stop 2 (SS2)
0008: Safe torque off (STO)
0009: Safe brake control (SBC)
000A: Diagnostic function
000B: Safety I/O-related observation

Remedy

- CPU may be faulty. Contact our service center.

V01	**Safe para storage memory err 1**	0004	

Details

Safety parameter (for internal processing) which is saved in the memory is illegal value.

Remedy

- Input Safety parameter file and turn power ON again.
- Clear the memory. (All data on the memory will be initialized. Back up the data as needed.)
- When the above action does not help restoring, memory may be faulty. Contact our service center.

V01	**Safe para storage memory err 2**	0005	

Details

Safety parameter which is saved in the memory is illegal value.

Remedy

- Input Safety parameter file and turn power ON again.
- Clear the memory. (All data on the memory will be initialized. Back up the data as needed.)
- When the above action does not help restoring, memory may be faulty. Contact our service center.

V01	**Safety initial process timeout**	0006	

Details

The initialization process of the safety function at power ON is not completed within specified time.

Remedy

- CPU may be faulty. Contact our service center.

V01	**NC-DRV initial safe comm error**	0007	Axis name

Details

The initial communication between NC unit and drive unit is incorrect. Displays the name of axis with error.

Remedy

- Check if there is no contact failure or no cable fracture after NC/Drive power OFF.
- NC unit or drive unit may be faulty. Contact our service center.

V01	**Safe IO init. process timeout**	0008	Unit info

Details

The safety I/O initialization process at power-up has not completed within the specified time.
Displays the information of the unit with an error.

bit24-27: Unit No.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Safety I/O unit may be faulty. Exchange the safety I/O unit.

V02	**Encoder error**	0001	Axis name

Details

The feedback position received from drive unit is incorrect.
Displays the name of axis with error.

Remedy

- Encoder may be faulty. Contact our service center.

V02	**NC-DRV safe communication err**	0004	Axis name

Details

The communication between NC unit and drive unit is incorrect.
Displays the name of axis with error.

Remedy

- Check if there is no contact failure or no cable fracture after NC/Drive power OFF.
- NC unit or drive unit may be faulty. Contact our service center.

V02	**Excess movement during pwr OFF**	0005	Axis name

Details

[Saved position at power shut OFF] and [restored position at power ON] are inconsistent in SLP/SCA encoder diagnosis during power OFF.
Displays the name of axis with error.

Remedy

There are two causes of this alarm; one is "the axis being moved during power OFF" and the other is "the incorrect restoring of the position at power ON".

- When it is likely with the cause "the axis being moved during power OFF", the alarm can be cancelled by turning ON the Safety reset signal while the Special safety alarm cancel signal is ON.
- When it is likely with the cause "the incorrect restoring of the position at power ON", encoder may be faulty. Contact our service center.

V03	Slave station comm. error 1	0001	Unit info

Details

The data received by the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped, and all the DOs of the safety I/O unit concerned are turned OFF.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

♦NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Slave station comm. error 2	0002	Unit info

Details

The data received by the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped, and all the DOs of the safety I/O unit concerned are turned OFF.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

♦NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Slave station comm. error 3	0003	Unit info

Details

The data received by the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped, and all the DOs of the safety I/O unit concerned are turned OFF.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

♦NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Slave station data compare err	0004	Unit info

Details

The data received by the safety I/O unit is inconsistent.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped, and all the DOs of the safety I/O unit concerned are turned OFF.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Check the user safety sequence circuit to see if the output signal control circuit is correct.
- Data corruption may have been caused due to noise. Take anti-noise measures on the connection between the NC unit and safety I/O unit.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Output OFF check error	0005	Unit info

Details

Output signal of the safety I/O unit fails to be OFF.
Displays the information of the unit with an error.

bit24-27: Unit No.

bit0-15: Signal BIT

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Output signal cross-check err	0006	Unit info

Details

Loop-back signals of the outputs from the safety I/O unit are inconsistent between PLC1 and PLC2.
Displays the information of the unit with an error.

bit24-27: Unit No.

bit0-15: Signal BIT

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Check the user safety sequence circuit to see if the output signal control circuit is correct.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Transmission cross-check error	0007	Unit info

Details

Output signals are inconsistent between the user safety sequence and safety I/O unit.
Displays the information of the unit with an error.

bit24-27: Unit No.

bit0-15: Signal BIT

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Check the user safety sequence circuit to see if the output signal control circuit is correct.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Reception cross-check error	0008	Unit info

Details

Input signals from the safety I/O unit are inconsistent between PLC1 and PLC2.
Displays the information of the unit with an error.

bit24-27: Unit No.

bit0-15: Signal BIT

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Any input device (emergency stop button, for example) connected to the safety I/O unit may be faulty. Check the input devices.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Host station comm. error 1	0009	Unit info

Details

The data received from the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Data corruption may have been caused due to noise. Take anti-noise measures on the connection between the NC unit and safety I/O unit.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Host station comm. error 2	0010	Unit info

Details

The data received from the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Host station comm. error 3	0011	Unit info

Details

The data received from the safety I/O unit is incorrect.
Displays the information of the unit with an error.

bit24-27: Unit No.

Safety observation target axes are stopped.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- The cable connecting between the NC unit and safety I/O unit may be disconnected or loosened. Check the cable.
- NC unit or safety I/O unit may be faulty. Exchange the NC unit or safety I/O unit.

V03	Drv safe receive crosscheck er	0012	ZR device No.

Details

The input signals from the drive's safety function are inconsistent.
The screen displays the No. of device ZR with an error.
Safety observation target axes are stopped.

Remedy

- NC unit may be faulty. Exchange the NC unit.

V03	User safety sequence 1 error	0013	Error cause number

Details

An error has occurred in User safety sequence 1.
The screen displays the error cause by the number.
Safety observation target axes are stopped.
User safety sequences 1 and 2 are both stopped.
All the DOs of the connected safety I/O unit are turned OFF.

Remedy

- Refer to the list of user safety sequence error details in Smart safety observation Specification manual. Cancel the error based on the displayed error cause, and then turn OFF and ON the NC power.

V03	User safety sequence 2 error	0014	Error cause number

Details

An error has occurred in User safety sequence 2. The screen displays the error cause.
Safety observation target axes are stopped.
User safety sequences 1 and 2 are both stopped.
All the DOs of the connected safety I/O unit are turned OFF.

Remedy

- Refer to the list of user safety sequence error details in Smart safety observation Specification manual. Cancel the error based on the displayed error cause, and then turn OFF and ON the NC power.

V03	**Output sig. cross check error**	0015	Unit info

Details

The signals output to the safety I/O unit are unmatched between PLC1 and PLC2.
Displays the information of the unit with an error.

bit24-27: Unit No.

bit0-15: Signal BIT

The axes covered by safety function come to a standstill.

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Check the user safety sequence to make sure the output signal control circuit is correct.
- Set tolerable time more than 300ms, when an output signal is controlled with 100ms timer.
- NC unit may be faulty. Replace the NC unit.

V04	**Safety observation & Smart both ON**	0001	

Details

The system has both an axis for which Safety observation is enabled (the parameter "#2313 SV113(SSF8)/bitF" or "#13229 SP229(SFNC9)/bitF" is "1") and an axis for which Smart safety observation is enabled (the parameter "#51101 SF_Disable" or "#51301 SF_SDisable" is "0").

Remedy

- Disable Safety observation for all the axes (Set the parameters "#2313 SV113(SSF8)/bitF" and "#13229 SP229(SFNC9)/bitF" to "0"), and turn ON the NC reset signal.
- Disable Smart safety observation for all the axes (Set the parameters "#51101 SF_Disable" and "#51301 SF_SDisable" to "1"), and turn the power OFF and ON.

V04	**Safety IO device unconnectable**	0002	Safety I/O unit-connected channel

Details

A safety I/O unit has been connected with the smart safety observation option OFF or with the parameters "#51101 SF_Disable" and "#51301 SF_SDisable" set to "1" for all the axes.The screen displays the channel to which the safety I/O unit is connected by the bit number.

bit0: Operation panel

bit1: RIO 1CH

bit2: RIO 2CH

bit3: RIO 3CH

bit4: RIO 4CH

Remedy

- Disconnect the safety I/O unit, and then turn OFF and ON the power.
- If you wish to use Smart safety observation, implement the following and then turn the power OFF and ON.

  - Enable the option.

  - Turn "0" the axis parameter of Smart safety observation ("#51101 SF_Disable" / "#51301 SF_SDisable").

V04	**Safe IO disabled: connect err**	0003	Safety I/O unit-connected channel

Details

A safety I/O unit has been connected to the I/O connection channel where RIO1.0 unit is connected.
The screen displays the I/O connection channel to which the safety I/O unit is connected using a bit number.

bit0: Operation panel

bit1: RIO 1CH

bit2: RIO 2CH

bit3: RIO 3CH

bit4: RIO 4CH

Remedy

- For the I/O connection channel where the safety signal input/output is conducted, no devices other than RIO2.0 unit or safety I/O unit can be connected.
- If you are unable to change the I/O device configuration immediately, set the parameters "#51101 SF_Disable" and "#51301 SF_SDisable" to "1" for all the axes, and turn OFF and ON the power. This prevents occurrence of this alarm.

V04	**Safe IO disabled: no safe I/Os**	0004	

Details

None of the I/O connection channels is connected to a safety I/O unit.

Remedy

- Connect a safety I/O unit to the I/O connection channel through which the safety signals are input/output.
- If you are unable to change the I/O device configuration immediately, set the parameters "#51101 SF_Disable" and "#51301 SF_SDisable" to "1" for all the axes, and turn OFF and ON the power. This prevents occurrence of this alarm.

V04	**Safety PLC is not yet written**	0005	

Details

Safety PLC has not been written.

Remedy

- Write safety PLC and turn the power OFF and ON.
- If safety PLC is not ready, set the parameters "#51101 SF_Disable" and "#51301 SF_SDisable" to "1" for all the axes, and then turn the power OFF and ON. This prevents occurrence of this alarm.

V04	**NC-DRV safety comm. Disabled**	0006	Optical channel No.

Details

The optical channel connected to an axis for which the parameter "#51101 SF_Disable" or "#51301 SF_SDisable" is set to "0" is configured with any drive unit other than MDS-E Series. (MDS-D Series drive unit is connected to the said channel.)
The screen displays the No. of optical communication channel of this error.

Remedy

- Do not connect any drive unit other than MDS-E Series to the optical channel of the axis for which the parameter "#51101 SF_Disable" or "#51301 SF_SDisable" is set to "0".
- If an MDS-E Series drive unit is not ready, set the parameters "#51101 SF_Disable" / "#51301 SF_SDisable" to "1" for all the axes of the said channel, and then turn OFF and ON the power.
  This prevents occurrence of this alarm.

V04	**EMG stop signal device illegal**	0007	Emergency stop device index No.

Details

The channel No. or station No. of the emergency stop signal device (set by parameters) does not coincide with any contact point (channel/station No. specified by the safety I/O assignment parameters RIO CH No and RIO Station No.) of the safety I/O unit. The screen displays the index No. of the incorrectly set emergency stop signal device.

0001: EMG_Dev1_ch to EMG_Dev1_bit

0002: EMG_Dev2_ch to EMG_Dev2_bit

Remedy

- Change the channel, station or bit No. of emergency stop signal device to be one of the contact points of the safety I/O unit. And then turn OFF and ON the power.
- When you change the setting of emergency stop signal device channel No. (EMG_Dev1_ch / EMG_Dev2_ch) to 0, and turn OFF and ON the power, the designation of emergency stop signal device is disabled, so this alarm is cleared.

V04	**Safe IO assign para setting er**	0009	Channel No.

Details

Safety I/O device assignment parameter is incorrect.

- The safety I/O device assignment parameter of the connected safety I/O unit is not set.
- The safety I/O device assignment parameter is set for any disconnected safety I/O unit.
- The set channel No. or station No. is overlapped.

Remedy

- Make sure which safety I/O unit is connected, set the safety I/O device assignment parameters, and then turn OFF and ON the power.

V05	**SLS speed error**	0001	Axis name

Details

During SLS observation, the command/FB speed has exceeded the safely-limited speed(*) and has not dropped below the speed limit within the SLS detection delay time (set by parameter).
The screen displays the name of axis of this error.

(*)Safely-limited speed = SLS speed tolerance x SLS speed override / 100

Remedy

- This error can be cancelled by the safety reset signal while motor speed is under SLS limit.
- If the SLS speed tolerance or SLS speed override (set by parameter) is lower than the assumption, change the parameter setting and turn OFF and ON the power.
- If the SLS detection delay time (set by parameter) is shorter than the assumption, change the parameter setting.
- Check the safety ladder to make sure that SLS speed tolerance and SLS speed override are changed in a timely manner.

V05	**SLS deceleration error**	0002	Axis name

Details

The command/FB speed has failed to decelerate to the safely-limited speed(*) or lower within a period of SLS deceleration observation time (set by parameter) after start of SLS observation.
The screen displays the name of axis of this error.

(*)Safely-limited speed = SLS speed tolerance x SLS speed override / 100

Remedy

- This error can be cancelled by the safety reset signal while motor speed is under SLS limit.
- If the SLS speed tolerance or SLS speed override (set by parameter) is lower than the assumption, change the parameter setting and turn OFF and ON the power.
- If the SLS deceleration observation time (set by parameter) is shorter than the assumption, change the parameter setting.

V05	**SLP position error**	0003	Axis name

Details

The command position/FB position has gone out of the SLP position tolerance range (set by parameter) during the SLP observation, and failed to return to the tolerance range within the SLP detection delay time (set by parameter).
The screen displays the name of axis of this error.

Remedy

- This error can be cancelled by the safety reset signal while the axis is in the SLP position range.
- If the axis is out of the SLP position range, deactivate SLP observation, cancel this error using the safety reset signal, and then move the axis to a safe position in a manual mode.
- If the safely-limited position range (specified by the SLP position tolerance parameters) is smaller than the assumption, change the parameter settings and turn OFF and ON the power.
- If the SLP detection delay time (set by parameter) is shorter than the assumption, change the parameter setting.
- Check the safety ladder to make sure that SLP position tolerance is changed in a timely manner.

V05	**SOS speed error**	0004	Axis name

Details

The command/FB speed, which had exceeded the SOS stop speed (set by parameter) during SOS, has failed to drop to the SOS stop speed or lower within the SOS_V detection delay time (set by parameter).
The screen displays the name of axis of this error.

*If the conditions of two or more SOS-related alarms are met at a time, the notification priority order is as follows: SOS position deviation error > SOS travel distance error > SOS speed error.

Remedy

- This error can be cancelled by the safety reset signal while In SOS stop is ON.
- If In SOS stop is OFF, deactivate SOS, cancel this error using the safety reset signal, and then move the axis to a safe position in a manual mode.
- If the parameter of SOS stop speed is lower than the assumption, change the setting and turn OFF and ON the power.
- If the SOS_V detection delay time (set by parameter) is shorter than the assumption, change the parameter setting.

V05	**SOS position deviation error**	0005	Axis name

Details

The position deviation (difference between the command and FB positions), which had exceeded the SOS position deviation tolerance (set by parameter) during SOS, has failed to reduce to the SOS position deviation tolerance or smaller within the SOS_PD detection delay time (set by parameter).
The screen displays the name of axis of this error.

*If the conditions of two or more SOS-related alarms are met at a time, the notification priority order is as follows: SOS position deviation error > SOS travel distance error > SOS speed error.

Remedy

- Refer to the corrective actions of "SOS speed error" for how to cancel this error.
- If the SOS position deviation tolerance (set by parameter) is smaller than the assumption, change the setting and turn OFF and ON the power.
- If the SOS_PD detection delay time (set by parameter) is shorter than the assumption, change the parameter setting.

V05	**SOS travel distance error**	0006	Axis name

Details

The command/FB travel distance, which had exceeded the SOS travel distance tolerance (+/-) (set by parameter) during SOS, has failed to reduce to the SOS travel distance tolerance (+/-) or smaller within the SOS_P detection delay time (set by parameter).
The screen displays the name of axis of this error.

*If the conditions of two or more SOS-related alarms are met at a time, the notification priority order is as follows: SOS position deviation error > SOS travel distance error > SOS speed error.

Remedy

- Refer to the corrective actions of "SOS speed error" for how to cancel this error.
- If the SOS travel distance tolerance (+/-) (set by parameter) is smaller than the assumption, change the setting and turn OFF and ON the power.
- If the SOS_P detection delay time (set by parameter) is shorter than the assumption, change the parameter setting.

V05	**SS1 deceleration error**	0007	Axis name

Details

The SS1 deceleration observation time (set by parameter) has elapsed with the command/FB speed exceeding the SOS stop speed (set by parameter) since the start of SS1.
The screen displays the name of axis of this error.

Remedy

- This error can be cancelled by the safety reset signal while In SS1 stop is ON.
- If the SS1 deceleration observation time (set by parameter) is shorter than the assumption, change the parameter setting.

V05	**SS2 deceleration error**	0008	Axis name

Details

The command/FB speed has been exceeding the SOS stop speed (set by parameter) for a period of SS2 deceleration observation time (set by parameter) since the start of SS2.
The screen displays the name of axis of this error.

Remedy

- Deactivate SS2, cancel this error using the safety reset signal, and then move the axis to a safe position in a manual mode.
- If the SS2 deceleration observation time (set by parameter) is shorter than the assumption, change the parameter setting.

V06	**Safety external EMG stop is ON**	0001	

Details

Emergency stop signal is OFF (open status), although Safety external emergency stop is enabled.

Remedy

- Ensure the safety of the machine, and then turn the emergency stop signal ON (close status).

V07	**DRV safe circuit error**	Error No.	Axis name

Details

The drive unit's internal safety circuit has caused abnormal operation.
The error number corresponding to the contents of abnormal operation appears.

Remedy

- Drive unit may be damaged. Replace the drive unit.

13.2 Smart Safety Observation Warning (V50/V51/V52/V53/V54)

V50	**SSM hysteresis setting error**	0001	Axis name

Details

The SSM hysteresis width (set by parameter) is greater than the SSM speed (set by parameter).
The screen displays the name of axis of this error.

Remedy

- Change the SSM hysteresis width parameter to be a smaller value than the SSM speed.

V50	**Safe absol. posn unestablished**	0002	Axis name

Details

- After enabling SLP/SCA (Parameter SLP_Enable/SCA_Enable is 1), the Safety absolute position has never been established.
- In SLP/SCA encoder diagnosis during power OFF, [saved position at power shut OFF] and [restored position at power ON] are inconsistent.

While this alarm is ON, SLP/SCA will not operate. Displays the name of axis of this error.

Remedy

- This alarm can be cancelled by turning ON the Safety absolute position check signal. Before turning ON the Safety absolute position check signal, however, move the axis by manual operation to the position where the coordinate value is clear (the position that is marked or the reference position etc.) (When in the relative position detection system, it requires to operate reference position return to establish the reference position.) then compare the actual position and the displayed position to confirm both positions are corresponding.

V51	**SBT start disabled**	0001	Factor No.

Details

When turning ON the SBT start signal (SBTSTEXm / SBTSTMOm), the test start condition are not met.
Displays the factor why the brake test start is impossible
* When there are multiple factors, the smaller item is shown.

0001: part systems in automatic operation
0002: not in in-position
0003: in servo OFF state.
0004: in current limit
0005: the secondary axis in the synchronous control
0006: in superimposition control
0007: in arbitrary axis exchange control
0008: in mixed control
0009: Parameter for SBT error
000A: reference position establishment incomplete
000B: applies exclusive control of SBT
000C: the secondary axis SBT disabled
000D: the secondary axis SBT start disabled
000E: the secondary axis in single method

Remedy

- Check the start-enabled condition of the brake test. Eliminate the cause of the alarm, then enable the SBT start signal to start the brake test.

V51	**SBT warning 1**	0002	Axis name

Details

The axis movement amount exceeded the tolerable value in external brake test pattern 1.
Displays the name of axis with the error.

Remedy

- Move the axis to safe position by manual operation. Take corrective action to the brake after the power OFF. Then turn power ON and carry out the brake test again. This alarm will be cancelled when the test completes normally.
- This alarm can be cancelled with the safety reset signal. The external brake SBT incomplete signal (SBTNFEXm), however, remains ON.

V51	**SBT warning 4**	0005	Axis name

Details

The axis movement amount exceeded the tolerable value in the test pattern 1 of motor brake test.
Displays the name of axis with the error.

Remedy

- Move the axis to safe position by manual operation. Take corrective action to the brake after the power OFF. Then turn power ON and carry out the brake test again. This alarm will be cancelled when the test completes normally.
- This alarm can be cancelled with the safety reset signal. The external brake SBT incomplete signal (SBTNFEXm), however, remains ON.
  When this alarm is cancelled with the safety reset signal, however, the motor brake SBT incomplete signal (SBTNFMOm) remains ON.

V51	**SBT warning 5**	0006	Axis name

Details

The axis movement amount exceeded the tolerable value in the test pattern 2 of motor brake test.
Displays the name of axis with the error.

Remedy

- Move the axis to safe position by manual operation. Take corrective action to the brake after the power OFF. Then turn power ON and carry out the brake test again. This alarm will be cancelled when the test completes normally.
- This alarm can be cancelled with the safety reset signal. The external brake SBT incomplete signal (SBTNFEXm), however, remains ON.
  When this alarm is cancelled with the safety reset signal, however, the motor brake SBT incomplete signal (SBTNFMOm) remains ON.

V51	**SBT warning 6**	0007	Axis name

Details

The axis movement amount exceeded the tolerable value in the test pattern 3 of motor brake test.
Displays the name of axis with the error.

Remedy

- Move the axis to safe position by manual operation. Take corrective action to the brake after the power OFF. Then turn power ON and carry out the brake test again. This alarm will be cancelled when the test completes normally.
- This alarm can be cancelled with the safety reset signal. The external brake SBT incomplete signal (SBTNFEXm), however, remains ON.
  When this alarm is cancelled with the safety reset signal, however, the motor brake SBT incomplete signal (SBTNFMOm) remains ON.

V52	**PLC safety stop is active**	0001	Axis name

Details

A PLC input signal "Safe stop 1 request" or "Safe torque off request" is OFF (normal close).
The screen displays the name of axis for which the said signal is OFF.

Remedy

- Ensure the safety of the machine, and then turn ON either Safe stop 1 request signal or Safe torque off request signal.

V53	**Warning on 24Hr continuous ON**	0001	Unit info

Details

Output signal of the safety I/O unit has been kept ON for 24 hours or longer.
Displays the information of the unit which is subject to the warning.

bit24-27: Unit No.

bit0-15: Signal BIT

(*) The unit No. indicates the unit in which an alarm occurs.
The channel No. and station No. to which the unit is connected can be confirmed by the following parameters.

Channel No.: #51501 + 10 × (Unit No. - 1)

Station No.: #51502 + 10 × (Unit No. - 1)

Remedy

- Turn OFF the output signal concerned through the user safety sequence, or use the output OFF check function to make sure that the output signal turns OFF.

V54	**Simple test mode is active**	0001

Details

- Smart safety observation target axis (the parameters "#51101 SF_Disable" = 0 / "#51301 SF_SDisable" = 0) is defined as a hypothetical axis (the parameter "#51015 safe_drv_test" = 1). In this case some alarms fail to occur, thus avoid this setting while a drive unit is being connected.
- NC system is set to a simulation mode (the parameter "#1168 test" = 1). During this mode some alarms fail to occur, thus do not use this mode while a safety I/O unit is connected.

Remedy

- Connect MDS-E Series drive to all the axes subject to safety observation, set the parameter ("#51015 safe_drv_test" = 0) and then turn OFF and ON the power.
- Connect a safety I/O unit, set the parameter ("#1168 test" = 0) and then turn OFF and ON the power.

Revision History

Revision Date	Manual No.	Changes
Aug. 2025	IB(NA)1501808-A	First edition created. Contents of "M800V/M80V Series Alarm/Parameter Manual" (IB-1501623-H) were divided as follows. ♦"M800V/M80V Series Alarm Manual" (IB-1501808-A) ♦"M800V/M80V Series Parameter Manual" (IB-1501809-A) Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B4. The following chapters were revised. - 1 Operation Errors (M) - 3 Servo/Spindle Alarms (S) - 4 MCP Alarms (Y) - 5 System Alarms (Z) Other mistakes were corrected.
Jan. 2026	IB(NA)1501808-B	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B8. Chapters 1, 4 and 12 were modified. The main changes are as follows: The following alarms were added. ♦M01 0018, M01 0310, M01 0380, M01 1160, M91 0002, M91 0003, M91 0045, M91 0046, M91 0089, M91 9000, M91 9001, M91 9002, M91 9003, M92 (1. Operation Errors (M)) The contents were revised as follows. ♦M01 1024 (1. Operation Errors (M)) ♦Y51 0108 (4. MCP Alarms (Y)) ♦P29 (12. Program Errors (P)) Other mistakes were corrected.
May 2026	IB(NA)1501808-C	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version BB. Chapters 1 and 12 were modified. The main changes are as follows. Chapter numbers are indicated in parentheses. The following alarms were added. ♦M01 1093, M01 1094, M01 1609 (1. Operation Errors (M)) ♦P490, P491, P492, P493, P494, P993, P994 (12. Program Errors (P)) The contents were revised as follows. ♦M01 0013 (1. Operation Errors (M)) ♦P49, P204 (12. Program Errors (P)) Other mistakes were corrected.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)
HQ and Central Region Service Center (Chicago)
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
Minneapolis, MN Service Satellite
Detroit, MI Service Satellite
Grand Rapids, MI Service Satellite
Cleveland, OH Service Satellite
St. Louis, MO Service Satellite
Indianapolis, IN Service Saltellite

South/East Region Service Center (Georgia)
1845 SATELLITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL +1-678-258-4529 / FAX +1-678-258-4519
Charleston, SC Service Satellite
Charlotte, NC Service Satellite
Raleigh, NC Service Satellite
Dallas, TX Service Satellite
Houston, TX Service Satellite
Hartford, CT Service Satellite
Knoxville, TN Service Satellite
Nashville, TN Service Satellite
Huntsville, AL Satellite
Pittsburg, PA Service Satellite
Tampa, FL Service Satellite
Miami, FL Satelite
Lafayette, LA Service Satellite
Allentown, PA Satelite

Western Region Service Center (California)
5900-B KATELLA AVE. - 5900-A KATELLA AVE. CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650
San Francisco, CA Satelite
Seattle, WA Service Satellite
Denver, CO Service Satellite

Canada Region Service Center (Toronto)
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Queretaro)
Parque Tecnológico Innovación Querétaro, Lateral Carretera Estatal 431, Km 2+200, Lote 91 Modulos 1 y 2
Hacienda la Machorra, CP 76246, El Marqués, Querétaro, México
TEL: +52-442-153-6050
Monterrey, NL Service Satellite
Mexico City, DF Service Satellite

BRAZIL

MITSUBISHI ELECTRIC DO BRASIL COMÉRCIO E SERVIÇOS LTDA.
Votorantim Office
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3023-9000
Blumenau, Santa Catarina Office

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
European Service Headquarters (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-5000 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)
SCHELMENWASENSTRASSE 16-20, 70567 STUTTGART, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center (Paris)
2 RUE DE L'UNION, 92565 RUEIL-MALMAISON CEDEX, FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)
240, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center (Milan)
VIA ENERGY PARK 14, VIMERCATE 20871 (MB) ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy Service Satellite (Padova)
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-288-780 / FAX: +44-1707-278-695

Spain Service Center
CTRA. RUBI, 76-80 8174 SAINT CUGAT DEL VALLES, BARCELONA, SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Hungary Service Center
MITSUBISHI ELECTRIC EUROPE B.V. HUNGARIAN BRANCH
BUDAÖRS OFFICE PARK, SZABADSÁG ÚT 117., 2040 BUDAÖRS, HUNGARY
TEL: +36-70-433-2263

Turkey Service Center
MITSUBISHI ELECTRIC TURKEY ELEKTRİK ÜRÜNLERİ A.Ş
SERIFALI MAHALLESI KALE SOKAK. NO.41 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-969-2500 / FAX: +90-216-661-44-47

Czech Republic Service Center
MITSUBISHI ELECTRIC EUROPE B.V. PRAGUE OFFICE
Pekařská 621/7, 155 00 PRAHA 5, CZECH REPUBLIC
TEL: +420-734-402-587

Sweden Service Center
HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251200 / FAX: +46-8-6251014

Bulgaria Service Center
AKHNATON Ltd. (Service Partner)
4 ANDREJ LJAPCHEV BLVD. POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine Service Center (Kiev)
CSC Automation Ltd. (Service Partner)
4 B, YEVHENA SVERSTYUKA STR., 02002 KIEV, UKRAINE
TEL: +380-44-494-3344 / FAX: +380-44-494-3366

South Africa Service Center
Adroit Technologies (Service Partner)
20 WATERFORD OFFICE PARK, WATERFORD DRIVE, CNR OF WITKOPPEN ROAD,
FOURWAYS JOHANNESBURG SOUTH AFRICA
TEL: +27-11-658-8100 / FAX: +27-11-658-8101

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

PHILIPPINES

MELCO FACTORY AUTOMATION PHILIPPINES INC.
Head Office
128 LOPEZ RIZAL STREET, BRGY., HIGHWAY HILLS, MANDALUYONG CITY , MM PHILIPPINES 1550
TEL: +63-2-8256-8042 / FAX: +632-8637-2294

Philippines Service Center
KM.23 WEST SERVICE ROAD SSH, CUPANG ,MUNTINLUPA CITY, PHILIPPINES
TEL: +63-2-8807-0420 / FAX: +63-2-8842-5202

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO., LTD.
Vietnam Ho Chi Minh Service Center
11TH & 12TH FLOOR, VIETTEL TOWER B, 285 CACH MANG THANG 8 STREET, WARD 12, DISTRICT 10,
HO CHI MINH CITY, VIETNAM
TEL: +84-28-3910-5945 / FAX: +84-28-3910-5947

Vietnam Hanoi Service Center
14TH FLOOR, CAPITAL TOWER, 109 TRAN HUNG DAO STREET, CUA NAM WARD,
HOAN KIEM DISTRICT, HA NOI CITY, VIETNAM
TEL: +84-24-3937-8075 / FAX: +84-24-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA
Indonesia Service Center (Cikarang)
JL. KENARI RAYA BLOK G2-07A, DELTA SILICON 5, LIPPO CIKARANG - BEKASI 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

MALAYSIA

MITSUBISHI ELECTRIC SALES MALAYSIA SDN. BHD.
Malaysia Service Center (Kuala Lumpur Service Center)
LOT 11, JALAN 219, P.O BOX 1036, 46860 PETALING JAYA, SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-7626-5032

Johor Bahru Service Center
9, Jalan Perjiranan 4/6, Bandar Dato Onn, 81100 Johor.
TEL: 07-3642250 / 2256

Pulau Pinang Service Center
25-G, Pusat Perniagaan Perdana Jaya, Jalan Permatang Rawa, 14000 Bukit Mertajam, Pulau Pinang
TEL: +60-4-510-1838 / Fax: +60-4-510-1835

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO., LTD.
Thailand Service Center (Bangkok)
101, TRUE DIGITAL PARK OFFICE, 5TH FLOOR, SUKHUMVIT ROAD, BANGCHAK, PHRA KHANONG,
BANGKOK, 10260 THAILAND
TEL: +66-2-092-8600 / FAX: +66-2-043-1231-33

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
Service Head office and South India Service Center (Bangalore)
CNC eXPerience Park, PLOT NO. 57/A, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
PLOT 517, GROUND FLOOR, UDYOG VIHAR PHASE-III, GURUGRAM 122008, HARYANA, INDIA
TEL : +91-124-463-0300
Ludhiana Service Satellite
Panthnagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite
Manesar Service Satellite

West India Service Center (Pune)
ICC-Devi GAURAV TECHNOLOGY PARK, UNIT NO.402, FOURTH FLOOR, NORTH WING,
SURVEY NUMBER 191-192 (P), NEXT to INDIAN CARD CLOTHING COMPANY Ltd,
OPP. VALLABH NAGAR, PIMPRI, PUNE- 411 018, MAHARASHTRA, INDIA
TEL : +91-20-6819-2274
Kolhapur Service Satellite
Aurangabad Service Satellite
Mumbai Service Satellite

West India Service Center (Ahmedabad)
204-209, 2ND FLOOR, 31FIVE, CORPORATE ROAD PRAHLADNAGAR,
AHMEDABAD -380015, GUJARAT, INDIA
TEL : + 91-79-6777-7888
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
CNC Call Center
TEL: +86-400-921-5130

Shanghai Service Center
NO.1386 HONG QIAO ROAD, CHANG NING QU, SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
Qingdao Service Center
Suzhou Service Center
Wuhan Service Center
Ningbo Service Center
Hefei Service Center
Beijing Service Center
Tianjin Service Center
Xian Service Center
Dalian Service Center
Chengdu Service Center

Shenzhen Service Center
LEVEL8, GALAXY WORLD TOWER B, 1 YABAO ROAD, LONGGANG DISTRICT,
SHENZHEN 518129, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8229-3686
Dongguan Service Center
Xiamen Service Center

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Seoul Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9631 / FAX: +82-2-3664-8668

Korea Daegu Service Center
2F KT BLDG. 8, HOGUK-RO, BUK-GU, DAEGU 41518, KOREA
TEL: +82-53-382-7401 / FAX: +82-53-382-7412

Korea Changwon Technical Center
#205, 888, CHANGWON-DAERO, SEONGSAN-GU, CHANGWON-SI,
GYEONGSANGNAM-DO, 51532, KOREA
TEL: +82-55-261-5559 / FAX: +82-55-261-5558

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO. 8-1, GONGYEQU 16TH RD., XITUN DIST., TAICHUNG CITY 40768 , TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
11F, NO.88, SEC.6, ZHONGSHAN N. RD., SHILIN DIST., TAIPEI CITY 11155, TAIWAN
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan Tainan Service Center
11F-1, NO.30, ZHONGZHENG S. RD., YONGKANG DIST., TAINAN CITY 71067, TAIWAN
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA PTY. LTD.
Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269/ FAX: +61-2-9684-7245

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible. Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

COPYRIGHT 2025-2026 MITSUBISHI ELECTRIC CORPORATION ALL RIGHTS RESERVED

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M800V/M80V Series
MODEL CODE	100-920
Manual No.	IB-1501808

IB(NA)1501808(ENG)-C(2605)MEE
Made in JAPAN

Specifications are subject to change without notice.